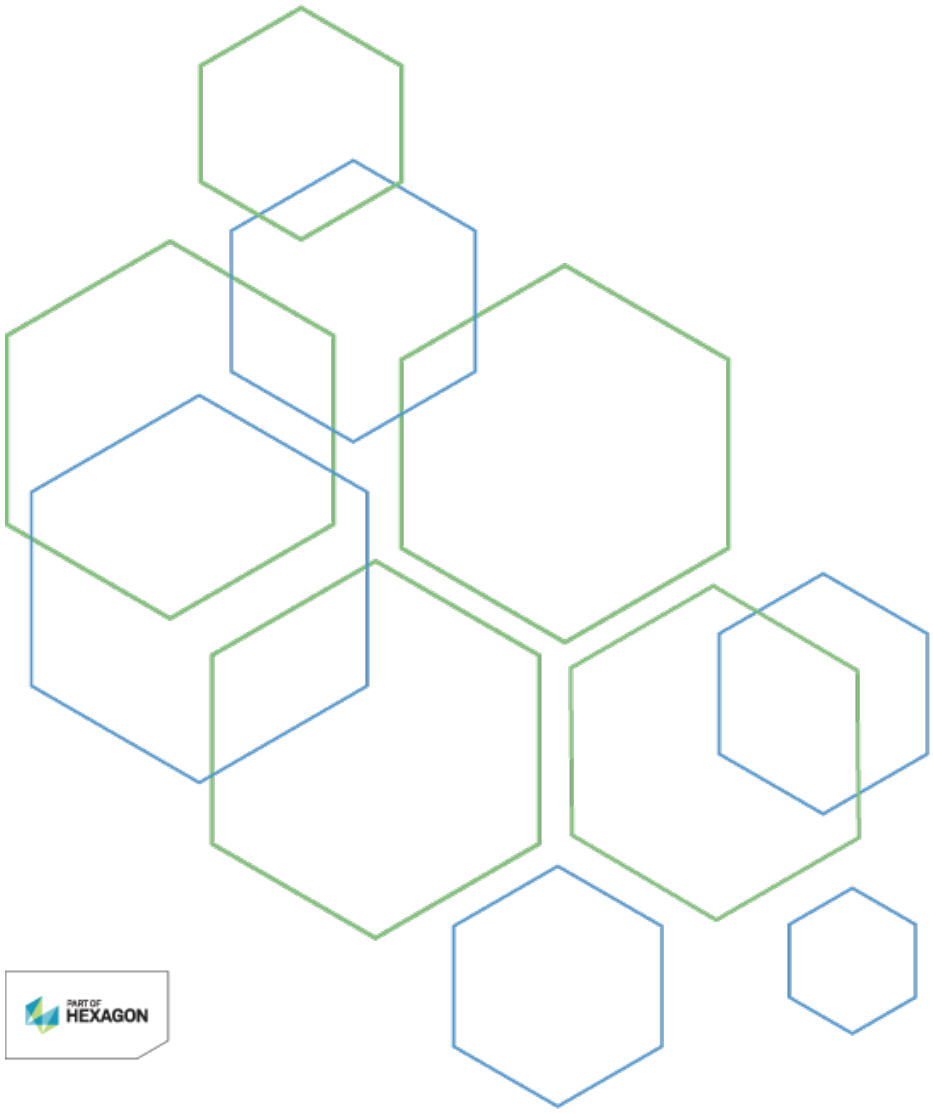
INTERGRAPH Smart 3D

Material Handling

User's Guide



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Contents

Preface

This document is a user's guide for the Material Handling functionality of Intergraph Smart™ 3D and provides command reference information and procedural instructions.

Documentation Comments

For the latest support information for this product, comments or suggestions about this documentation, and documentation updates for supported software versions, please visit *Intergraph Smart Support* (*https://smartsupport.intergraph.com*).

What's New in Material Handling

The following changes have been made to the Material Handling task.

Version 2016 (11.0)

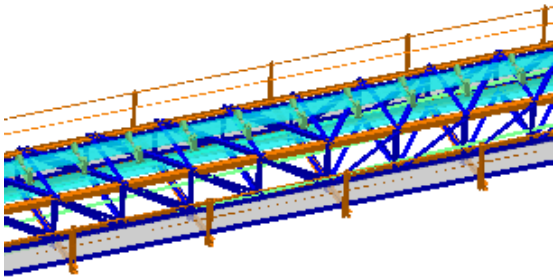
- Added new **Preview** and **Undo** options for placing fasteners. For more information, see *Place Fastener Openings Dialog Box* (on page 138). (P2 CP:290991)
- The **Quick Layout** command allows you to create a conveyor belt, including the belt profile, directly in the 3D view. For more information, see *Quick Layout* (on page 36). (P2 CP:241968)
- The **Place Belt Components in 3D** command allows you to create belt equipment (such as idlers and pulleys), modules (such as trusses), and trestles directly in the 3D environment. For more information, see *Place Belt Components in 3D* (on page 88). (P2 CP:2469183 and P2 CP:247957)
- The **Packing Thickness in 3D** command allows you to add or remove packing material thickness below idlers so that they mount perfectly on the trusses in the 3D environment. For more information, see *Packing Thickness in 3D* (on page 113). (P2 CP:250427 and P2 CP:250536)
- The **Create/Modify Belt Regions** command allows you to define flat, transition, and trough zones in each belt region. For more information, see *Create/Modify Belt Regions* (on page 52). (P2 CP:251147)
- A new option, **Materials Handling Wearplates**, has been added to **Select Catalog Object Type** list under **Catalog Browser** in the **Catalog Exporter** dialog box. For more information, see *Catalog Exporter Dialog Box* (on page 186). (P2 CP:268439)
- You can place overland conveyor belts with both horizontal and vertical curves. For more information, see *Place Conveyor Belt* (on page 23). (P2 CP:232441)
- Added a property called **IdlerHeight** to 2D belt component symbols. This property synchronizes the idler height value so that it displays appropriately in both 2D and 3D. For more information, see *Place module symbols in 2D* (on page 163). (P2 CP:247967 and P2 CP:252735)
- You can add custom commands as a toolbar in the Sketch 2D environment. For more information, see *Add custom commands to the Sketch 2D toolbar* (on page 161). (P2 CP:215587)
- You can now place a conveyor belt using an .xls file created by Sidewinder. For more information, see *Place a conveyor belt using an .xml or .xls file* (on page 26). (P3 CP:203907)
- Added new options, **Display Grid Lines**, **Footings**, and **Depth**, to the **Place 2D Trestles** dialog box. For more information, see *Place Tab (2D Trestles Dialog Box)* (on page 176). (P3 CP:218757) (P3 CP:245638)
- You can now select modules to edit in the graphic view. For more information, see *Modify a belt object's relationship to a belt* (on page 118). (P3 CP:257353)
- Added the **Zoom To** option to the **Belt Correction Tool** dialog box. For more information, see *Belt Correction Tool Dialog Box (Sketch 2D Custom Command)* (on page 171). (P3 CP:264356)

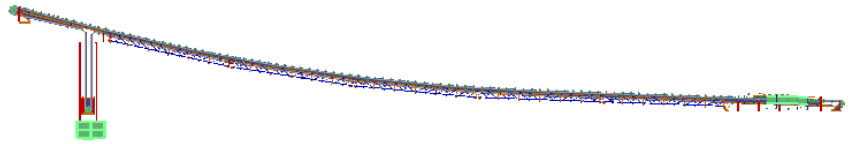
- Edited the **Preview** and **Finish** options on the **Edit Belt Region** dialog box. For more information, see *Create/Modify Belt Regions* (on page 52). (P3 CP:266602)
- SmartSketch is no longer required for the Material Handling task. (P3 CP:279797)
- Added the **Place Base Plate** custom command to place base plates from the catalog on specified columns. For more information, see *Place Base Plate* (on page 157). (P3 CP:281123)
- The **MHE Catalog Exporter** tool has been renamed as **Belt Components Catalog Exporter**. Access the tool from **Start > Intergraph Smart 3D > Database Tools > Belt Components Catalog Exporter**. For more information, see *Appendix: Belt Components Catalog Exporter* (on page 184). (P3 CP:283972)
- The **Modify Thickness Direction** command now aligns the primary profile orientation with the thickness direction. For more information, see *Modify Thickness Direction* (on page 148). (P4 CP:272539)
- Smart 3D can now calculate a fastener from the catalog that meets your requirements. For more information, see *Place Fastener Openings Dialog Box* (on page 138) and *Fastener Details Dialog Box* (on page 140). (P4 CP:230451 and CP:260277)

SECTION 1

Material Handling

The Material Handling task places and modifies conveyor systems that move material from one location to another. At the heart of the conveyor system is the belt which is the transfer medium for the material. Idlers, or rollers, are attached to trusses which support the belt. Transfer chutes help guide materials along the conveyor system. Use the Material Handling task to design and place these objects.





The Material Handling task has these commands:

	**Select** - Selects objects in the model. For more information, see *Selecting Objects* in the *Common User's Guide*.
	**Place Conveyor Belt** - Creates or modifies a conveyor belt from an XML file or Sketch 2D. For more information, see *Place Conveyor Belt* (on page 23).
	**Quick Layout** - Creates or modifies a conveyor belt directly in the 3D view. For more information, see *Quick Layout* (on page 36).
	**Create/Modify Belt Regions** - Creates or modifies region and zone properties on a conveyor belt. For more information, see *Create/Modify Belt Regions* (on page 52).
	**Truss Wizard** - Opens a wizard that steps you through the process of creating a new truss. For more information, see *Truss Wizard* (on page 59).
	**Place Belt Components** - Places multiple components along the selected belt profile. For more information see, *Place Belt Components* (on page 71).
	**Place Trestle** - Places a trestle in the model. For more information, see *Place Trestle* (on page 78).

	**Place Chute Shapes** - Places a chute shape in the model. For more information, see *Place Chute Shapes* (on page 82).
	**Place Belt Components in 3D** - Places belt equipment (such as idlers and pulleys), modules (such as trusses), and trestles directly in the 3D environment. For more information, see *Place Belt Components in 3D* (on page 88).
	**Packing Thickness in 3D** - Places additional packing material thickness below idlers so that they mount perfectly on the trusses. For more information, see *Packing Thickness in 3D* (on page 113).
	**Modify Belt Objects** - Modifies the position or orientation of the objects that have a parametric relation to a belt. For more information, see *Modify Belt Objects* (on page 118).
	**Create Control Points** - Creates multiple control points anywhere in the workspace, including grid intersections. For more information, see *Create Control Points* (on page 121).
	**Create Chute** - Places a chute in the model using control points as the vertices of the chute. For more information, see *Create Chute* (on page 126).
	**Place Fastener Openings** - Creates openings on chute flanges. For more information, see *Place Fastener Openings* (on page 132).
	**Measure Valley Angle** - Measures the valley angle at the intersection of two selected planes. For more information, see *Measure Valley Angle* (on page 142).
	**Create Volume View** - Creates a named view from a drawing volume. You use the named view to control the display of objects. For more information, see *Create Volume View* (on page 144).
	Export DSTV - Exports the DSTV file from a manufacturing XML file. This command is on the **File** menu. For more information, see *Export DSTV* (on page 153).
	Profile Auto Bound - Creates mutual bounding between stiffener profiles and profile edge reinforcements on mutually bound chute plates. This command is on the **Tools** menu. For more information, see *Profile Auto Bound* (on page 156).
	Modify Thickness Direction - Changes the thickness direction of the general plates of a chute. For more information, see *Modify Thickness Direction* (on page 148).
	Place Base Plate - Places base plates from the catalog on specified columns. This command is on the **Tools** menu. For more information, see *Place Base Plate* (on page 157).

SECTION 2

Material Handling Workflow

The following workflows are typically used in model creation. These workflows use multiple commands and move between Smart 3D Material Handling and Sketch 2D. Because your modeling requirements are unique, specific instructions are not always possible. However, enough detail is provided to get started and to guide you to more detailed procedures.

In This Section

Create a conveyor belt and equipment using 2D automation commands

This workflow uses **Place Conveyor Belt**  in Material Handling and Sketch 2D commands to model a conveyor belt that maintains the parametric relationship with belt components, such as equipment objects and modules.

Conveyor belt

1. Create a local coordinate system for the conveyor belt.

   Create a conveyor belt coordinate system (on page 26)

2. Create the path of the conveyor belt profile and place it in the model.

   Place a conveyor belt using an .xml or .xls file (on page 26)

   Place a conveyor belt using Sketch 2D (on page 27)

3. Define regions and zones for the belt.

   Define properties of a belt region (on page 53)

Add 2D automation commands

2D automation commands in Sketch 2D are used to create belt-related equipment. Before you use Sketch 2D for the first time, install the 2D automation custom commands delivered with Smart 3D Material Handling.

Add custom commands to the Sketch 2D toolbar (on page 161)

Belt equipment

1. Place symbols for belt equipment, such as pulleys and idlers, along the belt profile in Sketch 2D.

   Place equipment symbols in 2D (on page 162)

2. Place symbols for the conveyor belt support trusses in Sketch 2D.

   Place module symbols in 2D (on page 163)

3. Place symbols for miscellaneous objects, such as control points for creating chutes, in Sketch 2D.

   Place miscellaneous symbols in 2D (on page 164)

4. If required, associate idlers with the support trusses so that packing is calculated.

   Idler Packing Thickness (Sketch 2D - Custom Command) (on page 180)

Create a conveyor belt, equipment, and supports in 3D

This workflow uses commands in the 3D environment to model a conveyor belt, belt equipment, belt modules, and supporting structure. The belt maintains a parametric relationship with the belt components.

Create a conveyor belt

Create a conveyor belt coordinate system (on page 26)

Place a conveyor belt using an .xml or .xls file (on page 26)

Place a conveyor belt using Sketch 2D (on page 27)

Define properties of a belt region (on page 53)

Add belt equipment

Place idlers (on page 90)

Place a pulley (on page 91)

Add trusses to support belt equipment

Place modules (on page 92)

Add packing thickness in 3D (on page 114)

Add trestles to support the trusses

Place a trestle (on page 93)

Customize the conveyor profile and data report

This workflow is recommended for customizing the conveyor profile and data report using the labels functionality delivered with Smart 3D.

What do you want to do?

Add drawing custom attributes to the catalog

You must create a bulkload file that names each of the required custom attributes. The Excel workbook is then bulkloaded to the catalog.

1. Open the [*Product Folder*]\CatalogData\Bulkload\AdditionalFiles\Drawings-ExtendCustomAttributes.xls workbook.

   NOTE This workbook provides an example format for adding drawing custom attributes, such as custom drawing properties to be used in creating drawing borders. For more information, see *Drawings-ExtendCustomAttributes.xls Workbook* in the *Drawings and Reports Reference Data Guide*.

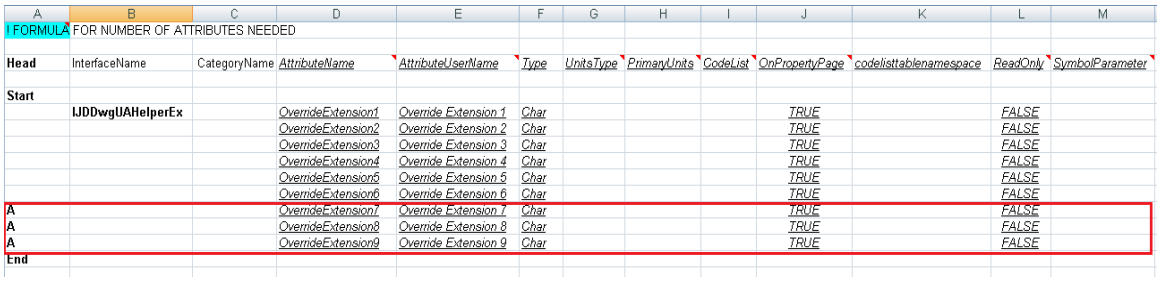
2. Select the **CustomInterfaces** sheet.
3. Select the row after the last row of attributes, and click **Insert > Row**.
4. In column **A**, type **A** in the first empty row to indicate a new drawing custom attribute is to be added during the next bulkload.

   NOTE The left-most column of the sheet (column **A**) specifies the bulkload action taken with regard to the row. For example, an **A** in the column indicates the drawing custom attribute is being added, a **D** specifies the attribute is deleted, and the letter **M** indicates the attribute is modified. An exclamation (**!**) symbol specifies that the row is ignored during bulkloading.

5. In the **AttributeName** column, type **OverrideExtension7**, and set **AttributeUserName** to **Override Extension 7**.

6. Repeat steps 3 through 5 to add two attributes: **OverrideExtension8** and **OverrideExtension9.**

   Your updated sheet should look similar to the example shown below.



	A	B	C	D	E	F	G	H	I	J	K	L	M
1	FORMULA	FOR NUMBER OF ATTRIBUTES NEEDED											
2													
3	Head	InterfaceName	CategoryName	AttributeName	AttributeUserName	Type	UnitsType	PrimaryUnits	CodeList	OnPropertyPage	codelisttablenamespace	ReadOnly	SymbolParameter
4													
5	Start												
6		IJDDwgUAHelperEx		OverrideExtension1	Override Extension 1	Char				TRUE		FALSE	
7				OverrideExtension2	Override Extension 2	Char				TRUE		FALSE	
8				OverrideExtension3	Override Extension 3	Char				TRUE		FALSE	
9				OverrideExtension4	Override Extension 4	Char				TRUE		FALSE	
10				OverrideExtension5	Override Extension 5	Char				TRUE		FALSE	
11				OverrideExtension6	Override Extension 6	Char				TRUE		FALSE	
12	A			OverrideExtension7	Override Extension 7	Char				TRUE		FALSE	
13	A			OverrideExtension8	Override Extension 8	Char				TRUE		FALSE	
14	A			OverrideExtension9	Override Extension 9	Char				TRUE		FALSE	
15	End												

7. Save the **Drawings-ExtendCustomAttributes.xls workbook**, and click **File > Close**.
8. Open the Bulkload utility, and define the settings as follows:
   a. Select **Append to existing catalog**.
   b. Clear the **Update Object Type Hierarchy** and **Catalog Views** boxes.

   TIP For detailed information about using the Bulkload utility, see *Loading Reference Data into the Catalog* in the *Smart 3D Reference Data Guide*.
9. Click **Load** to add the new reference data to the selected catalog.
10. When bulkload processing is complete, review the log file to ensure that there are no errors.

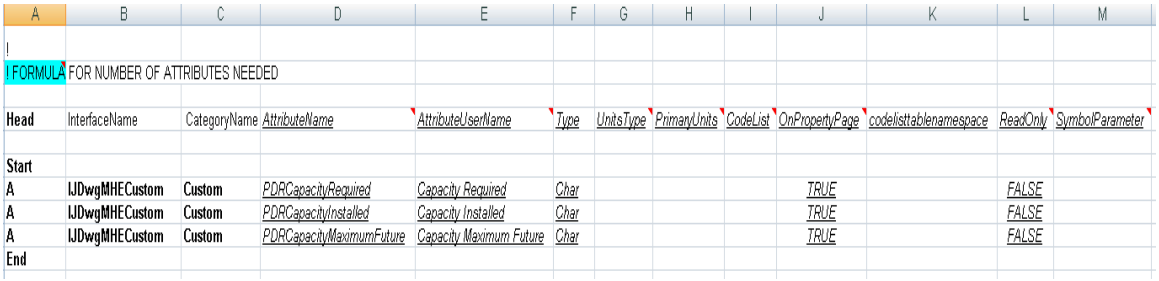
Define drawing custom attributes

1. Open the *[Product Folder]*\CatalogData\Bulkload\AdditionalFiles\Drawings-ExtendCustomAttributes.xls workbook, and save it as *MHPDRCustomAtt.xls* in the *[Product Folder]*\MaterialsHandling\CatalogData\Bulkload\DataFiles folder.
2. In the **MHPDRCustomAtt.xls** workbook, select the **CustomInterface** sheet, and do the following:
   a. Delete **IJDDwgUAHelperEx** in the **InterfaceName** column, and type **IJDwgMHECustom**.
   b. Type **Custom** in the **CategoryName** column.
   c. Delete the existing text in the **AttributeName** and **AttributeUserName** columns, and replace it with the attribute information listed below.

AttributeName	AttributeUserName
PDRCapacityRequired	Capacity Required
PDRCapacityInstalled	Capacity Installed
PDRCapacityMaximumFuture	Capacity Maximum Future

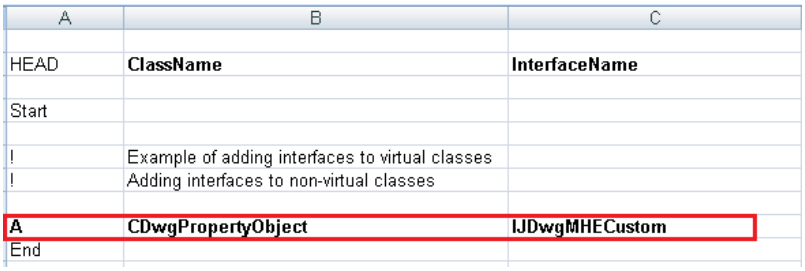
   d. In column **A**, type an **A** in each of the three rows to indicate a new drawing custom attribute is to be added during the next bulkload.

Your updated sheet should look similar to the example shown below.



	A	B	C	D	E	F	G	H	I	J	K	L	M
1	!												
2	! FORMULA	FOR NUMBER OF ATTRIBUTES NEEDED											
3													
4	Head	InterfaceName	CategoryName	AttributeName	AttributeUserName	Type	UnitsType	PrimaryUnits	CodeList	OnPropertyPage	codelisttablenamespace	ReadOnly	SymbolParameter
5													
6	Start												
7	A	IJDwgMHECustom	Custom	PDRCapacityRequired	Capacity Required	Char				TRUE		FALSE	
8	A	IJDwgMHECustom	Custom	PDRCapacityInstalled	Capacity Installed	Char				TRUE		FALSE	
9	A	IJDwgMHECustom	Custom	PDRCapacityMaximumFuture	Capacity Maximum Future	Char				TRUE		FALSE	
10	End												

3. Select the **CustomClassInterfaceList** sheet, and do the following:
   a. Set **ClassName** to **CDwgPropertyObject**.
   b. Set **InterfaceName** to **IJDwgMHECustom**.
   c. Type an **A** in column **A** to indicate a new drawing custom attribute class is to be loaded during the next bulkload.



	A	B	C
1			
2	HEAD	**ClassName**	**InterfaceName**
3			
4	Start		
5			
6	!	Example of adding interfaces to virtual classes	
7	!	Adding interfaces to non-virtual classes	
8			
9	**A**	**CDwgPropertyObject**	**IJDwgMHECustom**
10	End		

4. Save the **MHPDRCustomAtt.xls** workbook, and click **File > Close**.
5. Open the Bulkload utility, and do the following:
   a. Select **Append to existing catalog**.
   b. Clear the **Update Object Type Hierarchy** and **Catalog Views** boxes.

   TIP For detailed information about using the Bulkload utility, see *Loading Reference Data into the Catalog* in the *Smart 3D Reference Data Guide*.
6. Click **Load** to add the new reference data to the selected catalog.
7. When bulkload processing is complete, review the log file to ensure that there are no errors.

Create an XML schema file

The software uses the XML schema, (.XSD*)* file to populate the **Fields** list on the **Place Drawing Property Label** ribbon.

1. Open the *[Product Folder]*\3DRefData\SharedContent\Drawings\Catalog\Labels\Border\Schema\Configuration.xsd file using your default text editor, and save it as **MHECustom.xsd** in the same folder location.
2. In the element declaration on row 3, replace **Configuration** with *MHECustom*, so that the element name is defined as **<xs: element name="MHECustom">**.

3. Modify the next **element name** so that it defines one of the new drawing custom attributes in the **MHPDRCustomAtt.xls** workbook. For example, using PDRCapacityRequired, the line would read **<xs:element name="PDRCapacityRequired">**.
4. Modify the **pk name** to point to the **AttributeName** property of the bulkloaded attribute. For example, using the above drawing custom attribute name, the line would read **<pk name="PDRCapacityRequired"/>**.

   NOTE The **pk name** in the MHECustom.xsd file must match the related **AttributeName** property of the bulkloaded attribute.

   Your .xsd file should look similar to the example below:

```
<xs:element name="MHECustom">
    <xs:complexType>
        <xs:sequence>
            <xs:element name="PDRCapacityRequired">
                <xs:annotation>
                    <xs:appinfo>
                        <pk name="PDRCapacityRequired"/>
                    </xs:appinfo>
                </xs:annotation>
            </xs:element>
```

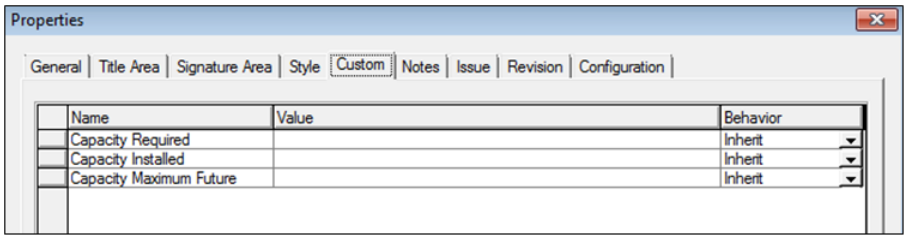
5. Continue modifying the **element name** and **pk name** strings to add the remaining two new drawing custom attributes to the .XSD file.

```
<xs:element name="MHECustom">
    <xs:complexType>
        <xs:sequence>
            <xs:element name="PDRCapacityRequired">
                <xs:annotation>
                    <xs:appinfo>
                        <pk name="PDRCapacityRequired"/>
                    </xs:appinfo>
                </xs:annotation>
            </xs:element>
            <xs:element name="PDRCapacityInstalled">
                <xs:annotation>
                    <xs:appinfo>
                        <pk name="PDRCapacityInstalled"/>
                    </xs:appinfo>
                </xs:annotation>
            </xs:element>
            <xs:element name="PDRCapacityMaximumFuture">
                <xs:annotation>
                    <xs:appinfo>
                        <pk name="PDRCapacityMaximumFuture"/>
                    </xs:appinfo>
                </xs:annotation>
            </xs:element>
```

6. Save the .XSD file, and exit the text editor.
7. Open the model, and create a drawing template in the Drawings and Reports task.

8. Right-click the drawing template in the **Management Console**, and select **Properties**.

   *The **Custom** tab displays the new drawing custom properties.*



Create the custom profile and data report

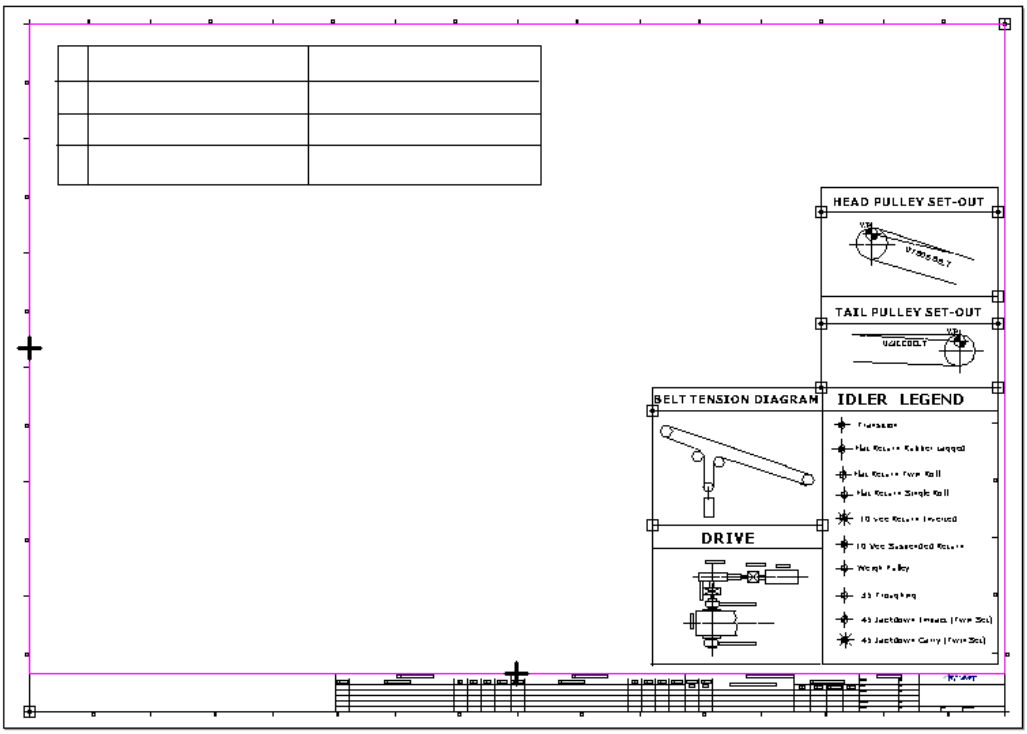
1. In the Drawings and Reports task, click **Tools > Edit Border Template**.

   *The **Select Template** dialog box displays.*

2. Select **BeltProfileData.sha** in the **Template** list, and click **OK**.

   *The template opens in the **Sketch 2D Drawing Editor.***

3. Click **Line** ⟋ on the vertical toolbar, and draw a report table in the drawing template similar to the one in the example below.



4. Click **Text Box**  on the vertical toolbar.

   *The **Text Box** ribbon appears.*

5. On the **Text Box** ribbon, set **Font Size** to **10mm**, and insert the required label text as shown in the example below.

1	Capacity Required	
2	Capacity Installed	
3	Capacity Maximum Future	

6. On the **Drawing Labels** toolbar, click **Place Drawing Property Label**  .

   *The **Place Drawing Property Label** ribbon appears.*

7. In the **Label Set** list, select **MHECustom**.

   TIP The list reflects the label sets within the drawing XML schema. The **Label Set** selection controls the contents of the **Fields** list and the enabling of other controls on the ribbon.

8. In the **Fields** list, select **PDRCapacityRequired** to use as your drawing property label.

9. Zoom in to the area of the border where you want to place the drawing property label, and click to place the label.

10. After the label is placed, use **Select**  on the vertical toolbar to position and resize the label as needed.

1	Capacity Required	PDRCapacityRequired

11. Repeat steps 6 through 10 to place and position the **PDRCapacityInstalled** and **PDRCapacityMaximumFuture** property labels, as shown in the example below.

1	Capacity Required	PDRCapacityRequired
2	Capacity Installed	PDRCapacityInstalled
3	Capacity Maximum Future	PDRCapacityMaximumFuture

12. Save the changes to the template, and exit the **Sketch 2D Drawing Editor**.

13. Create a composed drawing.

    TIP For detailed information about composed drawings, see *Composed Drawings* in the *Orthographic Drawings User's Guide*.

14. Right-click the composed drawing, and select **Properties**.

15. On the **Custom** tab, define the attributes as listed below and then click **OK**.

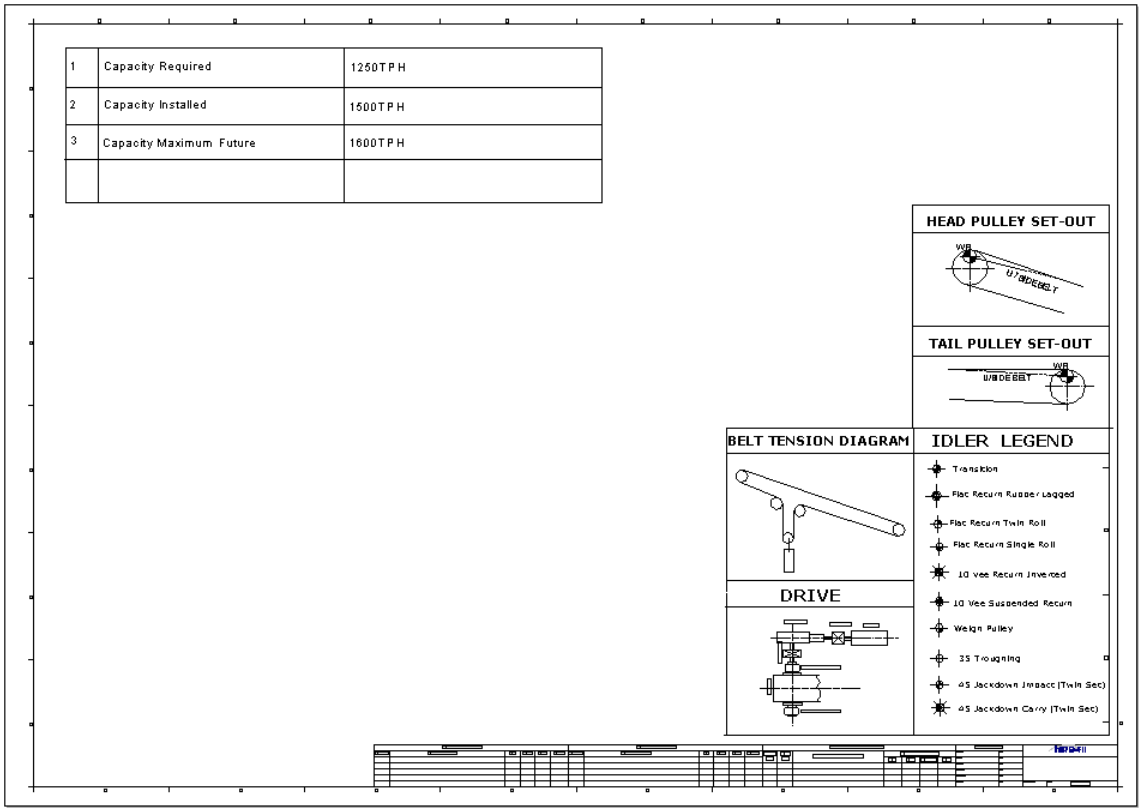
Name	Value	Behavior
Capacity Required	1250TPH	Override
Capacity Installed	1500TPJ	Override
Capacity Maximum Future	1600TPH	Override

16. Right-click the composed drawing, and select **Update Now**.

    TIP For detailed information about updating drawings, see *Update Now* in the *Orthographic Drawings User's Guide*.

17. Right-click the composed drawing, and select **Edit**.

    *The drawing opens in the **Sketch 2D Drawing Editor**, and displays the updated data.*



SECTION 3

Place Conveyor Belt

Creates or modifies a conveyor belt from an .xml file, an .xls file, or Sketch 2D. Conveyor belts created using this command maintain a parametric relationship with the belt components.

If you modify a belt that was used by **Place Belt Components (on page 71), Place Trestle** (on page 78), or **Place Belt Components in 3D** (on page 88), the software creates a parametric relationship between the belt and the components. For example, if you move the belt, the belt components also move to maintain their relationship to the belt.

Overland conveyor belts require horizontal and vertical curves. When you define overland conveyors in Sketch 2D, Smart 3D provides a **Profile** tab to display the conveyor belt design in the X-Z plane and a **Run** tab to display the design in the X-Y plane. Both sheets must pass through the origin (0,0). You can also import overland conveyor belts from .xml files.

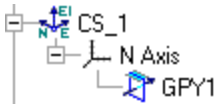
Place Conveyor Belt Ribbon

Belt Properties

Displays the **Conveyor Belt Properties** dialog box, in which you can view properties for the conveyor belt. For more information, see *Conveyor Belt Properties Dialog Box* (on page 29).

Select Coordinate System

Specifies an existing plant coordinate system to associate with the conveyor belt. The coordinate system must have, at a minimum, one Y plane at the origin.



Select Path Options

Specifies the path creation method. Click either **Path by Sketch 2D** or **Path by Import XML/XLS** to indicate how the conveyor belt is defined.

Finish

Places the conveyor belt.

Path by Sketch2D

Indicates that the conveyor belt is placed from Sketch 2D.

Path by Import XML/XLS

Indicates that the conveyor belt is placed from an .xml or .xls file.

Select Parent

Specifies the parent system for the conveyor belt. Click **More** to display the **Select System** dialog box.

Belt Designation

Specifies the material, default width, and default thickness of the belt. The software delivers the following belt designations:

- **ST1250_NN_600X6** - ST1250 nylon belt with a 600mm default width and a 6mm default thickness.
- **ST1251_EP_800X8** - ST1251 elastopolymer belt with an 800mm default width and an 8mm default thickness.
- **ST1252_PU_1000X10** - ST1252 polyurethane belt with a 1000mm default width and a 10mm default thickness.
- **ST1253_ST_1200X12** - ST1253 steel cord belt with a 1200mm default width and a 12mm default thickness.

All belt designations have a 3mm top cover thickness and a 2mm bottom cover thickness.

Belt Width

Specifies the width of the conveyor belt. You can change the default value.

Belt Thickness

Specifies the thickness of the conveyor belt. You can change the default value.

Name

Displays the name of the conveyor belt.

Export to XML

Displays the **Save As** dialog box so that you can export the conveyor belt profile to an .xml file. This option is only available when you are modifying an existing conveyor belt.

Import File Path

Specifies the path to the .xml or .xls file that contains the conveyor belt definition. Click **Browse** to browse to the file.

2D Sketcher Options

NOTE Some 2D Sketcher options listed below may not be applicable for the active command.

Sketching Plane

Specifies the sketching plane. This is the first step in defining the object.

Add Intersecting Item

Allows you to select objects that intersect the sketching plane in the 3D environment. You see the selected objects in the Sketch 2D environment when you are drawing.

Add Projection Item

Allows you to select objects that do not intersect the sketching plane in the 3D environment. The objects are projected onto the sketching plane, and you see the selected objects in the Sketch 2D environment when you are drawing.

Sketch 2D

Opens the Sketch 2D environment in which you can draw.

Coincident Plane

Specifies that you want to sketch on the plane that you select.

Offset from Plane

Specifies a sketching plane that is offset from a plane that you select. If you choose this option, you must define the offset distance.

Angle from plane

Specifies a sketching plane that is at a specified angle from a plane that you select. If you choose this option, you must define an axis of rotation and the angle or slope.

Plane by Point and Vector

Specifies the sketching plane using two points to define a vector normal to the sketching plane and a third point to define the sketching plane position along the vector.

Plane by Three Points

Specifies the sketching plane using three points that you specify in the model.

Plane by Vectors Normal

Specifies the sketching plane as being normal to another plane that you select and having a rotation parallel to a vector that you define.

Conveyor Belt Properties Dialog Box (on page 29)

What do you want to do?

- *Create a conveyor belt coordinate system* (on page 26)
- *Place a conveyor belt using an .xml or .xls file* (on page 26)
- *Place a conveyor belt using Sketch 2D* (on page 27)
- *Modify a 2D conveyor belt* (on page 28)
- *Export a conveyor belt to XML* (on page 28)
- *Delete a conveyor belt* (on page 39)

Create a conveyor belt coordinate system

You must use a plant coordinate system to place a conveyor belt.

1. In the Grids task, click **Grid Wizard** on the vertical toolbar.

   *The **Grid Wizard** dialog box appears.*

2. In the **Name** box, select or type a name.
3. In the **Type** box, select **Grids**.
4. In the **Axis for bearing** box, select an axis such as **East (X)** or **North (Y)**. This axis represents the plane of the belt profile.
5. In the **Bearing** box, type a value to refine the direction, in the format *N 0.00 deg E*.
6. In the **Origin** frame, set coordinates for an origin point. The origin must be on the belt centerline plane (along the Y-axis). A belt work point or pulley origin is usually appropriate.
7. Click **Next**.
8. On each of the remaining panels of the **Grid Wizard** dialog box, create X, Y, and Z planes using the following conventions:
   - In the **Reference CS** box, select the name created in the **Name** box on the first panel.

     IMPORTANT Do not select **Global**.
   - In the **Name rule** box, select **Position**.
   - Create a plane at 0.00 for each axis.
   - Create at least one additional plane along the positive axis of the X and the Y axes.

NOTES

- Geographic designations on axes, such as East (X) and North (Y), are plant conventions, and have no meaning for Material Handling.
- Z is used as the elevation axis.
- For more information, see *Grid Wizard* in the *Grids User's Guide*.

Place a conveyor belt using an .xml or .xls file

1. In the Material Handling task, click **Place Conveyor Belt** on the vertical toolbar.

   *The **Place Conveyor Belt** ribbon displays with the **Select Coordinate System** command activated.*

2. Select the plant coordinate system to associate with the conveyor belt from the **Workspace Explorer**.

   TIP You must select a plant coordinate system. Ship coordinate systems are not accepted for conveyor belt creation.

   The software prompts you to select a path option.

3. Click **Path by Import XML/XLS**.

4. Click **Browse** to navigate to the .xml or .xls file that defines the conveyor belt.

   *The software displays a **Browse Import File** dialog box.*

5. Select the file to import, and click **Open**.

   NOTE Use the drop-down menu to the right of the **File name** box to select a Sidewinder file.

   *The .xml or .xls file name is displayed in the **Import File Path** box.*

   TIP The .xls file must be created in Sidewinder. The .xml file must be created in a third-party application, such as Helix or Sidewinder, or by exporting a conveyor belt to .xml. For more information, see *Export a conveyor belt to XML* (on page 28).

6. Specify the parent object for the conveyor belt in the **Select Parent** box.
7. Specify the belt part for the conveyor belt in the **Belt Designation** box.
8. Specify the conveyor belt dimensions in the **Belt Width** and **Belt Thickness** boxes.
9. Click **Finish**.

   The conveyor belt displays in the model.

Place a conveyor belt using Sketch 2D

1. In the Material Handling task, click **Place Conveyor Belt** on the vertical toolbar.

   *The **Place Conveyor Belt** ribbon displays with the **Select Coordinate System** command activated.*

2. Select the plant coordinate system to associate with the conveyor belt from the **Workspace Explorer**.

   TIP You must select a plant coordinate system. Ship coordinate systems are not accepted for conveyor belt creation.

   The software prompts you to select a path option.

3. Click **Path by Sketch 2D**.
4. Specify the parent object for the conveyor belt in the **Select Parent** box.
5. Specify the belt designation for the conveyor belt in the **Belt Designation** box.
6. Specify the conveyor belt dimensions in the **Belt Width** and **Belt Thickness** boxes.
7. Click **Add Intersecting Item**, and select any intersecting items you need in Sketch 2D.
8. Click **Sketch 2D**.

   Sketch 2D opens and displays the selected objects.

   TIPS For overland conveyor belts:

   - The **Profile** tab displays the model in the X-Z plane.
   - The **Run** tab displays the model in the X-Y plane.
   - The **Profile** and **Run** sheets must both pass through the origin (0,0).

9. Create the belt profile and run using the drawing commands in Sketch 2D, or by pasting the profile geometry from an external SmartSketch file.

10. Click **Finish** in Sketch 2D.
11. Click **Finish** on the **Place Conveyor Belt** ribbon.

    The conveyor belt displays in the model.

Modify a 2D conveyor belt

1. Select a conveyor belt that was generated from XML or Sketch 2D.

   TIP You can select the conveyor belt from either the graphic view or **Workspace Explorer**.

   *The **Place Conveyor Belt** ribbon displays.*
2. Click **Select Path Options** .
3. To modify the conveyor belt by importing an XML file, click **Path by Import XML** .
   a. Change the conveyor belt parameters to meet your requirements.
   b. If necessary, set **Import File Path** to a different XML file.
4. To modify the conveyor belt by Sketch 2D, click **Path by Sketch 2D**
   a. Click **Add Intersecting Item** , and select any intersecting items you need in Sketch 2D.
   b. Click **Sketch 2D** .

      Sketch 2D opens and displays the selected objects.
   c. Modify the belt profile using the drawing commands in Sketch 2D or by pasting the profile geometry from an external SmartSketch file.
   d. Click **Finish** in Sketch 2D.
5. Click **Finish** on the **Place Conveyor Belt** ribbon.

   The software modifies the conveyor belt in the model.

Export a conveyor belt to XML

1. Select the conveyor belt to export.

   TIPS
   - You can select the conveyor belt from either the graphic view or **Workspace Explorer**.
   - You can select a conveyor belt created in XML, or a conveyor belt generated in Sketch 2D.
2. Click **Export to XML**.

   *The software displays the **Save As** dialog box.*
3. Specify the name and location for the XML file.
4. Click **OK**.

   The software exports the conveyor belt profile to an XML file.

Conveyor Belt Properties Dialog Box

Specifies the properties for the conveyor belt profile system that you are editing.

General Tab (Conveyor Belt Properties Dialog Box) (on page 29)

Report Data Tab (Conveyor Belt Properties Dialog Box) (on page 31)

Relationship Tab (on page 32)

Configuration Tab (on page 33)

Notes Tab (on page 34)

Topics

General Tab (Conveyor Belt Properties Dialog Box)

Displays belt properties that you can edit or that are automatically determined by the software at placement. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one conveyor belt, only the common properties between the selected belts display.

The following properties display for the delivered reference data.

Category

Displays the properties that you want to view for the belt. Belt properties have the following categories: **Standard** and **Responsibility**.

Standard

Parent System

Specifies the system to which the belt belongs.

Naming Rule

Specifies the naming rule that you want to use to name this belt.

- **Default Name Rule** - Names the belt using the format "BeltSystem-<*location*>-<*index*>" where <*location*> is the global workshare location ID and <*index*> is a unique index number that starts at 0001. For example, BeltSystem-1-0043.
- **Unique Name Rule** - Names the belt using the format "<*system*>-BeltSystem-<*location*>-<*index*>" where <*system*> is the name of the parent system to which the belt system belongs, <*location*> is the global workshare location ID and <*index*> is a unique index number that starts at 0001. For example, Conveyor-BeltSystem-1-0043.
- **User Defined** - Select to specify the belt system name yourself in the **Name** box.

Name

Displays the name of the belt. The belt name is based on the **Naming Rule** selection. If you want to type a new name for the belt, in the **Name Rule** box, select **User Defined**, and then type a name for the belt in the **Name** box.

Belt Type

Specifies the belt type.

Belt Width

Specifies the width of the belt.

Belt Thickness

Specifies the thickness of the belt

Top Cover Thickness

Specifies the thickness of the top cover.

Bottom Cover Thickness

Specifies the thickness of the bottom cover

Belt Material

Specifies the material used for the construction of the belt.

Belt Material Grade

Specifies the grade of the material used for the belt.

Belt Density

Specifies the density of the belt material.

Belt Designation

Specifies the load bearing capacity of the belt.

Belt Troughing Angle

Specifies the troughing angle of the belt path on the carry side. This troughing holds the material.

Belt Tail Transition Length

Specifies the length of the tail pulley side of the belt path.

Belt Head Transition Length

Specifies the length of the head pulley side of the belt path.

Responsibility

Cleaning Responsibility

Specifies the cleaning responsibility for the conveyor belt.

Design Responsibility

Specifies the design responsibility for the conveyor belt.

Fabrication Responsibility

Specifies the fabrication responsibility for the conveyor belt.

Installation Responsibility

Specifies the installation responsibility for the conveyor belt.

Painting Responsibility

Specifies the painting responsibility for the conveyor belt.

Requisition Responsibility

Specifies the requisition responsibility for the conveyor belt.

Supply Responsibility

Specifies the supply responsibility for the conveyor belt.

Testing Responsibility

Specifies the testing responsibility for the conveyor belt.

Report Data Tab (Conveyor Belt Properties Dialog Box)

Displays properties that you can edit or that are automatically determined by the software at placement. These properties are used for displaying Drawings and Reports data.

Category

Displays the properties that you want to view for the belt. Belt properties have one category: **Standard**.

Standard

Belt Speed

Specifies the speed of the belt.

Belt Speed Future

Specifies the future speed of the belt.

Belt Head Pulley T1

Specifies the tension of the belt at the tight side of the head pulley.

Belt Head Pulley T2

Specifies the tension of the belt at the slack side of the head pulley.

Belt Head Pulley TE

Specifies the effective tension of the belt at the head pulley.

Belt Tail Pulley T1

Specifies the tension of the belt at the tight side of the tail pulley.

Belt Tail Pulley T2

Specifies the tension of the belt at the slack side of the tail pulley.

Belt Tail Pulley TE

Specifies the effective tension of the belt at the tail pulley.

Required Capacity

Specifies the required capacity of the belt.

Installed Capacity

Specifies the installed capacity of the belt.

Max Future Capacity

Specifies the maximum future capacity of the belt.

Belt Lift

Specifies the lift distance associated with the belt.

Horizontal Centers

Specifies the distance between the centers of the head pulley and the tail pulley.

Conveyed Material

Specifies the material conveyed on the belt.

Conv Max Bulk Density

Specifies the bulk density of the conveyed material.

Moisture Content

Specifies the moisture content of the conveyed material.

Angle of Repose

Specifies the angle of repose for the conveyed material.

Belt Rated Mass

Specifies the rated mass of the conveyor belt.

Surcharge Angle

Specifies the surcharge angle of the conveyed material.

Total Belt Length

Specifies the total length of the belt.

Relationship Tab

Displays all objects related to the selected object for which you are viewing properties. For example, if you are viewing the properties of a pipe run, the related pipeline, features, parts, associated control points, hangers or supports, and equipment display on this tab. All WBS assignments, including project relationships, appear on this tab.

Additional examples for marine relationships are as follows:

- For plate and profile system properties, the related bounded objects, bounding objects, and connections are shown.
- For plate and profile system part properties, parent systems are shown.

- For assembly connection properties, all connected objects are shown.
- For the properties of a frame connection on a member, supported, supporting, and auxiliary supporting parts are shown.
- For split connection properties, the parent and auxiliary supporting parts are shown.

Name

Specifies the name of the object.

Type

Specifies the type of object. To change the options on the list, edit the **Weld Type** select list in Catalog.

Go To

Displays the properties of the selected object.

Configuration Tab

Displays the creation, modification, and status information about an object.

NOTE You cannot define the filters using the **Configuration** tab.

Plant

Displays the name of the model. You cannot change this value.

Permission Group

Specifies the permission group to which the object belongs. You can select another permission group, if needed. Permission groups are created in Project Management.

Transfer

Reassigns ownership of the selected model objects from their current permission group to another satellite or host permission group. This option is only available if the active model or project is replicated in a workshare configuration. The option is not available if all of the objects in the select set already belong to another location and are non-transferable. For more information, see *Transfer Ownership Dialog Box* in the *Common User's Guide*.

NOTE The **Transfer** option does not apply to the filters and surface style rules.

Approval State

Specifies the current status of the selected object or filter. The display depends on your access level. You might be unable to change the status of the object. The list is defined by the ApprovalStatus codelist.

NOTE You can only edit or manipulate an object with a status of **Working**.

Status

Specifies the location of the object in the workflow process. Changing this property sets the **Approval State**. The list is controlled by the ApprovalReason codelist in the ApprovalReason.xls file. You must bulkload this file. For more information, see *ApprovalReason* in the *Reference Data Guide*.

Date Created

Specifies the creation date of the object.

Created by

Specifies the name of the person who created the object.

Date Last Modified

Specifies the date when the object was last modified.

Last Modified by

Specifies the name of the person who last modified the object.

Transfer Ownership Dialog Box

Allows you to specify a new location and permission group for the selected model objects.

Current location

Displays the name of the location with which the current permission group is associated. All of the objects in the select set must belong to the same location.

Current permission group

Displays the name of the permission group with which the selected objects are currently associated. If all of the objects in the select set do not belong to the same permission group, this box appears blank.

New location

Specifies the name of the location to which you want to assign the objects. In a global workshare configuration, this box lists all the locations in which you have write access to one or more permission groups. The selection in this box filters the entries in the **New permission group** box.

New permission group

Specifies the new permission group to which to assign the selected objects. If you specify a value in the **New location** box, this list displays all permission groups to which you have write access in the selected location. If you do not specify a value in the **New location** box, this list includes all permission groups to which you have write access in all locations except the current location. This box is blank if you do not have write access to any permission groups at any locations other than the current one.

NOTE We strongly recommend that administrators follow naming convention rules that include the location as a prefix in the permission group name.

Notes Tab

Creates and edits user-definable text placed by the designer on an object in the model. The notes provide special instructions related to the object for the fabricator and are available in downstream tasks. For example, the notes appear in two-dimensional drawings and within design review sessions.

NOTE Only one note of a given kind from a given object can be shown on a drawing. For example, if there are two fabrication notes on a piping part, then only one of the notes shows on the drawing. It is important to know about and to consider this situation when defining notes on an object in the modeling phase. For example, you can display one Fabrication note and one Installation note by defining two separate labels for the two kinds of notes.

Key point

Specifies the key point on the object to which you want to add a note.

Notes at this location, listed by name

Lists all notes for the selected key point on the object.

Date

Displays the date that the note was created. The system automatically supplies the date.

Time

Displays the time that the note was created. The system automatically supplies the time.

Purpose of note

Specifies the purpose of the note.

Author

Displays the login name of the person who created the note. The system automatically supplies this information. You cannot change this information.

Note text

Defines the note text. The software does not limit the length of the note text.

Show dimension

Indicates that the note generates a dimension.

If you are displaying the properties for a Support component, then a dimension can be included for the component in the Support drawings, if you select the **Show dimension** option. The note must be associated with one of the key points for the Support component. It is recommended that you set the **Purpose of note** as **Fabrication**, but this is not a requirement. The note **Name** and **Note text** are not used when you select this option.

New Note

Creates a new note on the object.

Standard Note

Displays a list of standard notes from which you can select. This feature is not available in this version.

Highlight Note

Highlights the note in the graphic view so that you can easily find the note and the object to which it is related. This feature is not available in this version.

Delete Note

Deletes the currently displayed note.

SECTION 4

Quick Layout

Creates or modifies a conveyor belt directly in the 3D view. You can define the parameters of the belt profile.

NOTE If you modify a belt that was used by **Place Belt Components (on page 71), Place Trestle** (on page 78), or **Place Belt Components in 3D** (on page 88), the software creates a parametric relationship between the belt and the components. For example, if you move the belt, the belt components also move to maintain their relationship to the belt.

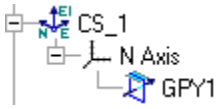
Quick Layout Ribbon

Belt Properties

Displays the **Conveyor Belt Properties** dialog box, in which you can view properties for the conveyor belt. For more information, see *Conveyor Belt Properties Dialog Box* (on page 29).

Select Coordinate System

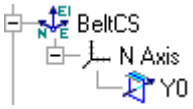
Specifies an existing plant coordinate system to associate with the conveyor belt. The coordinate system must have, at a minimum, one Y plane at the origin.



Select 2 points to place Coordinate system

Defines a new coordinate system by selecting two points in the graphic view. The first point defines the origin, and the second point defines the direction of the positive E-axis (the Y-axis). The software defines the EL-axis (the Z-axis) as up.

NOTE The direction of the belt is along the positive E-axis. The software creates the coordinate system **BeltCS** with one Y plane at the origin.



Sketch Belt in 3D

Allows you to layout the belt profile directly in the 3D model by selecting a geometric construction and defining the belt parameters. This option opens the **Belt Path Palette** dialog box and the **Geometric Construction Explorer** dialog box.

Belt Path Palette Dialog Box (on page 39)

Geometric Construction Explorer (on page 50)

Finish

Places the conveyor belt.

Select Parent

Specifies the parent system for the conveyor belt. Click **More** to display the **Select System** dialog box.

Belt Designation

Specifies the material, default width, and default thickness of the belt. The software delivers the following belt designations:

- **ST1250_NN_600X6** - ST1250 nylon belt with a 600mm default width and a 6mm default thickness.
- **ST1251_EP_800X8** - ST1251 elastopolymer belt with an 800mm default width and an 8mm default thickness.
- **ST1252_PU_1000X10** - ST1252 polyurethane belt with a 1000mm default width and a 10mm default thickness.
- **ST1253_ST_1200X12** - ST1253 steel cord belt with a 1200mm default width and a 12mm default thickness.

All belt designations have a 3mm top cover thickness and a 2mm bottom cover thickness.

Belt Width

Specifies the width of the conveyor belt. You can change the default value.

Belt Thickness

Specifies the thickness of the conveyor belt. You can change the default value.

Name

Displays the name of the conveyor belt.

Continue

Places a coordinate system defined with **Select 2 points to place Coordinate system** . This option is only available when creating a new belt.

Convert to 2D

Converts the belt to a belt with a Sketch 2D belt profile. After converting, you can modify the belt using **Place Conveyor Belt** , providing more flexibility than **Quick Layout** . For more information, see *Place Conveyor Belt* (on page 23). **Convert to 2D** is only available when modifying a belt.

Belt Path Palette Dialog Box (on page 39)

What do you want to do?

- *Create a conveyor belt coordinate system* (on page 26)
- *Place a conveyor belt in 3D* (on page 38)
- *Modify a 3D conveyor belt* (on page 38)
- *Convert a 3D conveyor to 2D* (on page 39)
- *Delete a conveyor belt* (on page 39)

Place a conveyor belt in 3D

1. In the Material Handling task, click **Quick Layout** on the vertical toolbar.

   The command ribbon displays.

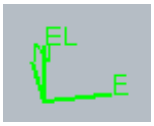
2. Select or create a parent system.
3. Select an existing plant coordinate system.

   -OR-

   Create a new coordinate system.

   a. Click **Select 2 points to place Co-ordinate system**.

   b. Select a point for the origin, and then select a point in the direction of the Y-axis.

   c. Click **Continue**.

4. Select a belt designation, and change the belt width and thickness if required.
5. Click **Sketch Belt in 3D**.

   *The **Belt Path Palette** dialog box and the **Geometric Construction Explorer** dialog box display.*

6. Follow the help for the selected belt layout macro. For more information, see *Belt Path Palette Dialog Box* (on page 39).

Modify a 3D conveyor belt

1. Click **Select** on the vertical toolbar.
2. Select **Conveyor Belt** in the **Locate Filter** box.
3. Select a conveyor belt created using **Quick Layout**.

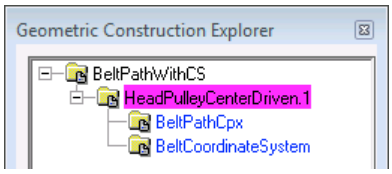
   TIP You can select the conveyor belt from either the graphic view or **Workspace Explorer**.

   *The **Quick Layout** ribbon displays.*

4. Click **Sketch Belt in 3D**.

   *The **Belt Path Palette** dialog box, **Geometric Construction Explorer** dialog box, and the belt layout macro ribbon display.*

5. Select the belt macro in the **Geometric Construction Explorer**.



The ribbon for the macro displays.

6. Change the belt parameters on the ribbon as required.

   The preview displays the belt with the new parameters.

7. Click **Continue**, and then click **Close**.

   The ribbon display returns to the **Quick Layout** *ribbon.*

8. Change the parent system, belt designation, belt width, and belt thickness as required.
9. Click **Finish**.

   The software updates the belt.

Convert a 3D conveyor to 2D

1. Click **Select** on the vertical toolbar.
2. Select **Conveyor Belt** in the **Locate Filter** box.
3. Select a conveyor belt created using **Quick Layout**.

   TIP You can select the conveyor belt from either the graphic view or **Workspace Explorer**.

   The ***Quick Layout*** *ribbon displays.*

4. Click **Convert to 2D**.

   The ribbon display returns to the **Select** *ribbon.*

5. Select the conveyor belt again.

   The **Place Conveyor Belt** *ribbon displays.*

6. You can now modify the belt using **Place Conveyor Belt**.

   Modify a 2D conveyor belt (on page 28)

Delete a conveyor belt

1. Click **Select** on the vertical toolbar.
2. Select **Conveyor Belt** in the **Locate Filter** box.
3. Select the conveyor belt to delete.
4. Click **Delete**.

   The software deletes the belt. The coordinate system used to create the belt remains.

Belt Path Palette Dialog Box

Specifies the 3D belt profile layout. The software creates a belt profile by using a geometric construction macro. The **Belt Path Palette** dialog box displays when you click **Sketch Belt in 3D** on the **Quick Layout** ribbon. After you select a macro, the appropriate options display on the ribbon.

Filters

Limits the available macros on a tab to the selected type.

Clear Recent

Removes recently selected macros from the **Recent** tab.

Help

Displays help for the selected macro in a preview window. If no help is available, a preview of the selected macro displays.

NOTE You can also select **Tools** > **Automatic Preview** to display the preview window.

	**Tail Pulley Center Driven** - Creates a belt profile with the center of the tail pulley at the origin of the selected coordinate system. For more information, see *Tail Pulley Center Driven* (on page 41).
	**Head Pulley Center Driven** - Creates a belt profile with the center of the head pulley at the origin of the selected coordinate system. For more information, see *Head Pulley Center Driven* (on page 45).

NOTE The **Recent** tab displays macros recently selected on the other tabs. Click **Clear Recent** to remove macros from this tab.

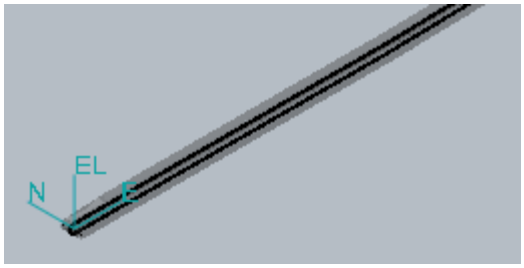
The **Geometric Construction Explorer** also displays. As you create the belt path, a list of belt macros and output geometry display.

Geometric Construction Explorer (on page 50)

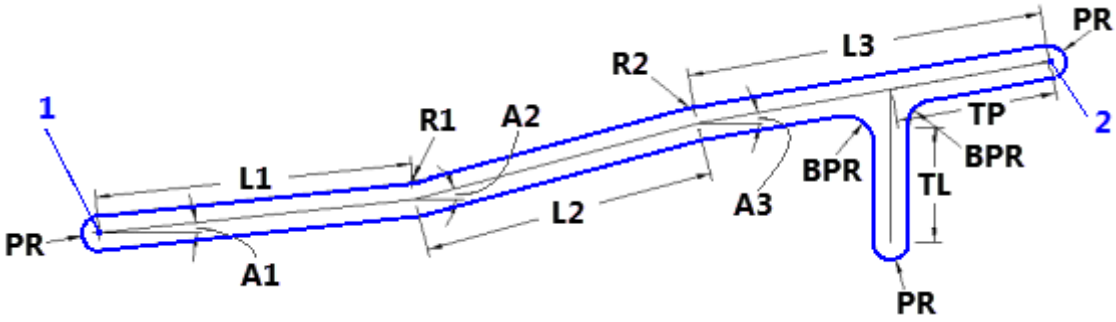
Tail Pulley Center Driven

Tooltip: TailPulleyCenterDriven

Creates a belt profile with the center of the tail pulley at the origin of the selected coordinate system. The belt extends along the positive E-axis (the Y-axis).



Required Input	**Output**
Coordinate system **Parameters** See the graphic below.	Belt profile curve



1 - Tail pulley

2 - Head pulley

L1, **L2**, **L3** - Length of segment

R1 - Concave radius

R2 - Convex radius

BPR - Bend pulley radius

PR - Pulley radius

TL - Take-up length

TP - Take-up position

A1, **A2**, **A3** - Inclination of segment

Tail Pulley Center Driven Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1

Specifies the coordinate system used.

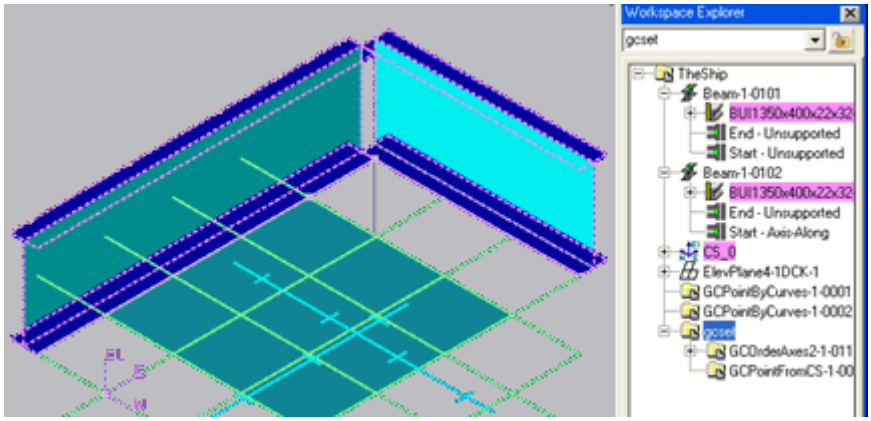
Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.

- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

L1

Defines the length of the first belt segment.

L2

Defines the length of the second belt segment.

L3

Defines the length of the third belt segment.

R1

Defines the concave radius between the first and second belt segments.

R2

Defines the convex radius between the second and third belt segments.

BPR

Defines the bend pulley radius at the belt take-up.

PR

Defines the radius of the head, tail, and take-up pulleys.

TL

Defines the length of the belt take-up.

TP

Defines the distance of the take-up from the head pulley.

A1

Inclination of the first belt segment.

A2

Inclination of the second belt segment.

A3

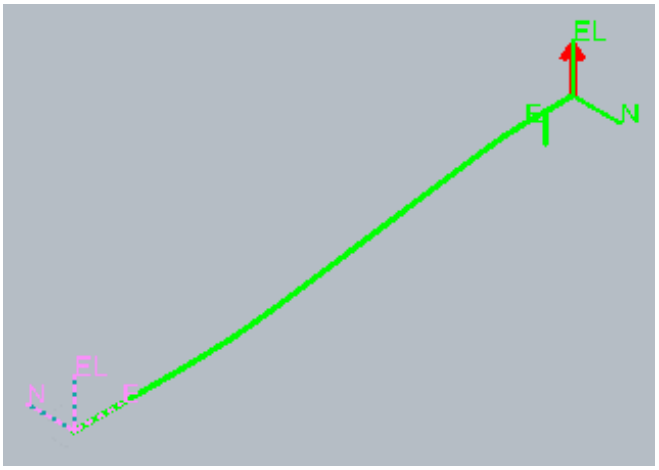
Inclination of the third belt segment.

NOTE For more information on the belt profile parameters, see the graphic displaying above.

Place a belt profile with the tail pulley at the coordinate system origin

1. Click **TailPulleyCenterDriven** on the **Belt Path Palette**.
2. Select the coordinate system to use as the origin for the tail pulley.

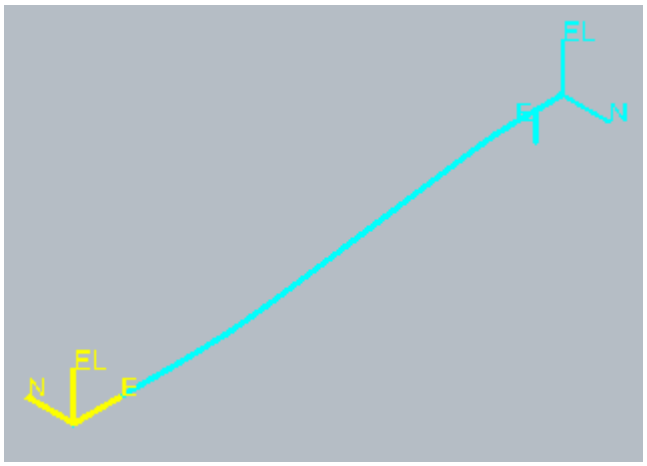
   A preview of the belt profile displays. The direction of the belt is along the positive E-axis.



3. Type any required changes to the belt parameters.

4. Click **Continue**.

   The preview colors change.

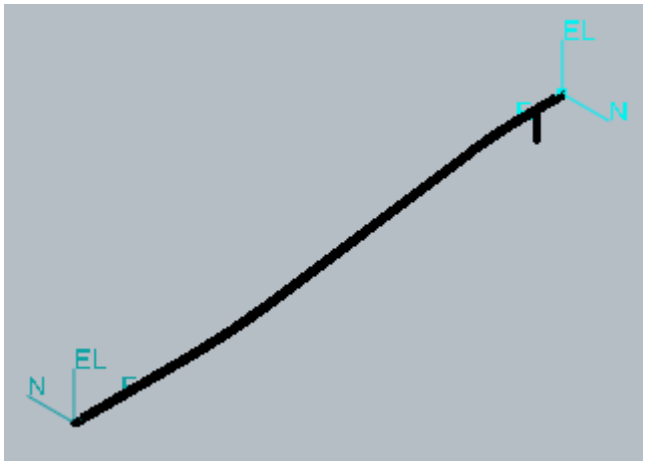


5. Click **Close**.

   *The ribbon returns to the **Quick Layout** ribbon.*

6. Click **Finish**.

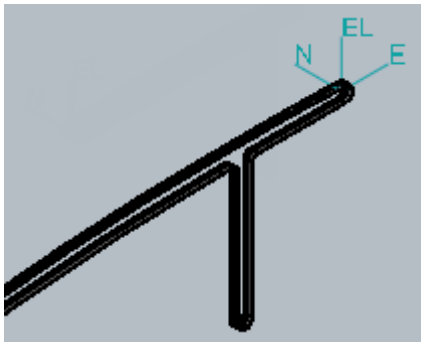
   The software creates the belt.



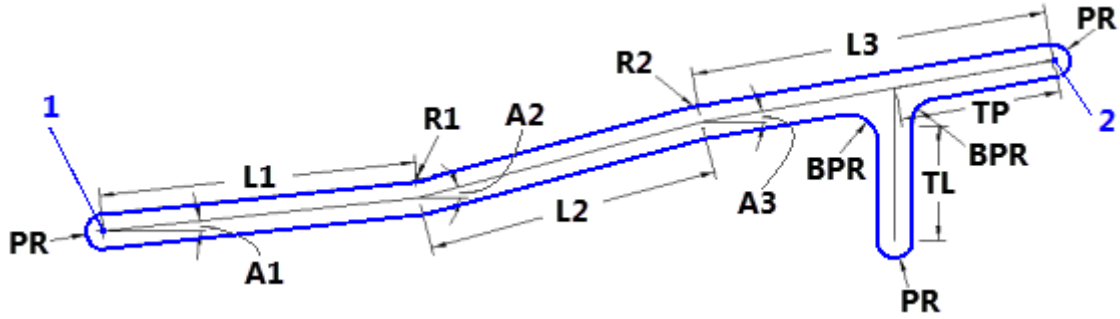
Head Pulley Center Driven

Tooltip: HeadPulleyCenterDriven

Creates a belt profile with the center of the head pulley at the origin of the selected coordinate system. The belt extends along the positive E-axis (the Y-axis).



Required Input	Output
Coordinate system **Parameters** See the graphic below.	Belt profile curve



1 - Tail pulley

2 - Head pulley

L1, **L2**, **L3** - Length of segment

R1 - Concave radius

R2 - Convex radius

BPR - Bend pulley radius

PR - Pulley radius

TL - Take-up length

TP - Take-up position

A1, **A2**, **A3** - Inclination of segment

Head Pulley Center Driven Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1

Specifies the coordinate system used.

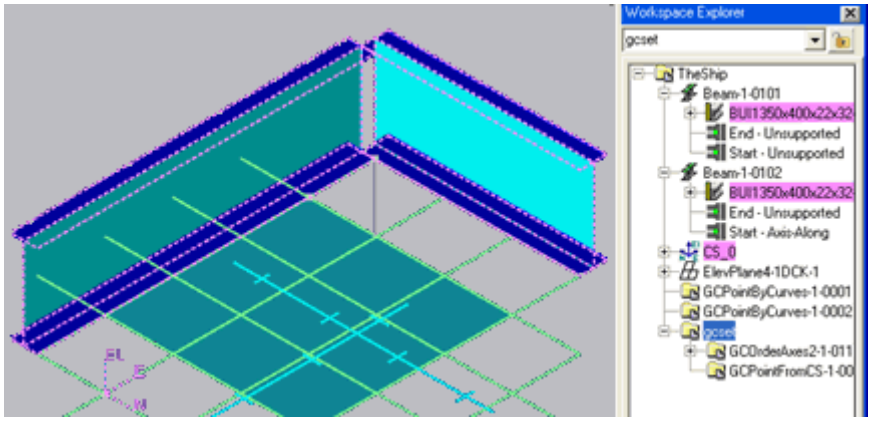
Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.

- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

L1

Defines the length of the first belt segment.

L2

Defines the length of the second belt segment.

L3

Defines the length of the third belt segment.

R1

Defines the concave radius between the first and second belt segments.

R2

Defines the convex radius between the second and third belt segments.

BPR

Defines the bend pulley radius at the belt take-up.

PR

Defines the radius of the head, tail, and take-up pulleys.

TL

Defines the length of the belt take-up.

TP

Defines the distance of the take-up from the head pulley.

A1

Inclination of the first belt segment.

A2

Inclination of the second belt segment.

A3

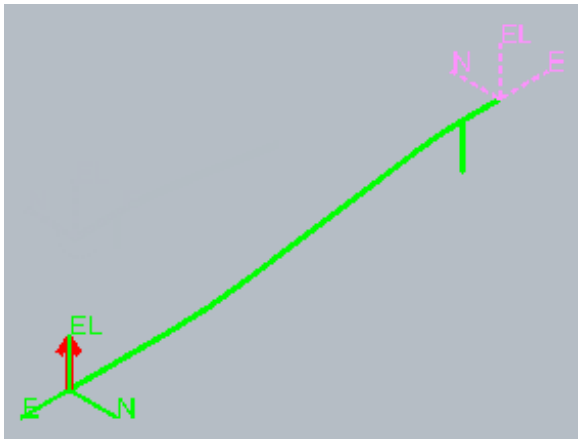
Inclination of the third belt segment.

NOTE For more information on the belt profile parameters, see the graphic displaying above.

Place a belt profile with the head pulley at the coordinate system origin

1. Click **HeadPulleyCenterDriven** on the **Belt Path Palette**.
2. Select the coordinate system to use as the origin for the tail pulley.

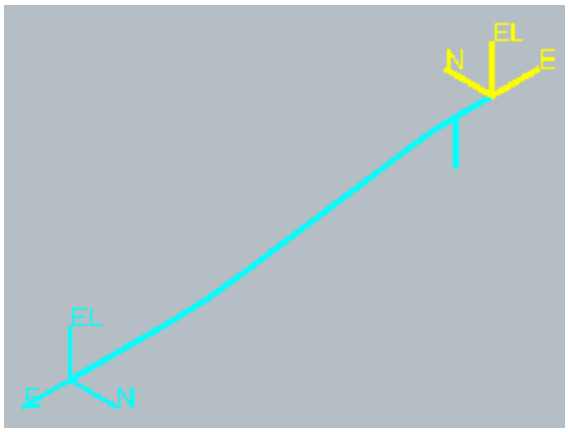
   A preview of the belt profile displays. The direction of the belt is along the positive E-axis.



3. Type any required changes to the belt parameters.

4. Click **Continue**.

   The preview colors change.

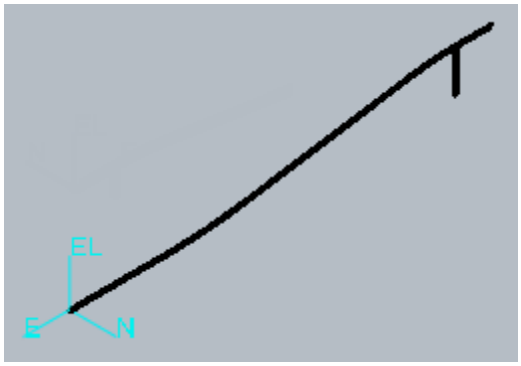


5. Click **Close**.

   *The ribbon returns to the **Quick Layout** ribbon.*

6. Click **Finish**.

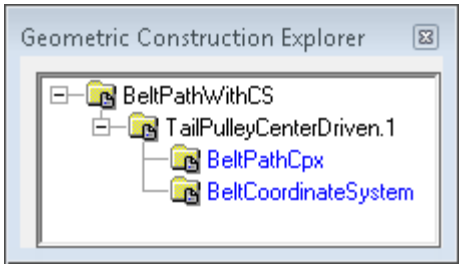
   The software creates the belt.



Geometric Construction Explorer

Displays geometry related to a belt profile. The geometry is added after you define the belt parameters on the ribbon and click **Continue**.

The **Geometric Construction Explorer** displays parameters in a hierarchical list similar to the **Workspace Explorer**. Selecting a blue belt geometry name highlights the geometry in the graphic view.



Commands

The following commands are available by right-clicking an item in the hierarchical list:

Show

Turns on the graphical display of a hidden object in the graphic view.

Hide

Turns off the graphical display of an object in the graphic view. The names of hidden objects display in italics in the **Geometric Construction Explorer**.

Export

Exports the selected geometric construction to an XML file. This command is only available when you right-click the top level geometric construction set.

Import

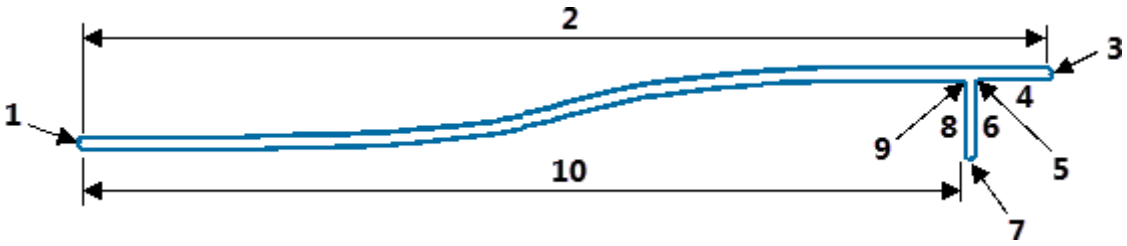
Imports a previously exported geometric construction from an XML file into the workspace. The imported geometric construction replaces any existing geometric construction. This command is only available when you right-click the top level geometric construction set.

NOTE **Export** and **Import** are primarily intended for use by administrators creating complex geometric construction macros. The commands can also be used to save multiple geometric constructions, such as a set of seams or profile stiffener landing curves on a hull, that you may not complete in a single session, or that you want to reuse in a similar model.

SECTION 5

Create/Modify Belt Regions

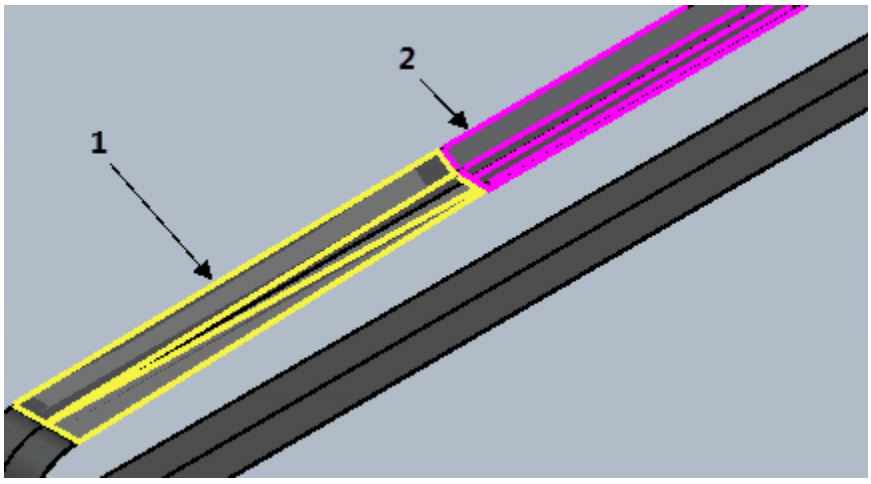
Creates or modifies region and zone properties on a conveyor belt. **Place Conveyor Belt** defines belt regions automatically, but the software hides regions until you use **Create/Modify Belt Regions**. Regions are defined at each pulley and at each belt section between two pulleys. Regions numbers increase in the direction of belt motion, starting with the tail pulley.



1 - Tail pulley region
2 - Carry region
3 - Head pulley region
5, **7**, **9** - Pulley regions
4, **6**, **8**, **10** - Belt regions

If you do *not* specify region properties using **Create/Modify Belt Regions**, then the belt displays default cross-section geometry: flat for the return side of the belt, and the following zones for the carry region:

- Transition (at the tail end)
- Trough
- Transition (at the head end)



1 - Transition
2 - Trough

When you use **Create/Modify Belt Regions**, you can specify properties for a series of zones within the selected belt region.

- Flat
- Transition
- Trough
- Transition
- Trough
- Transition
- Flat

If you do not need a zone, you can set length of the zone to zero.

NOTE You can only select belt regions. If you select a pulley region, a warning message displays.

Edit Belt Region Dialog Box (on page 55)

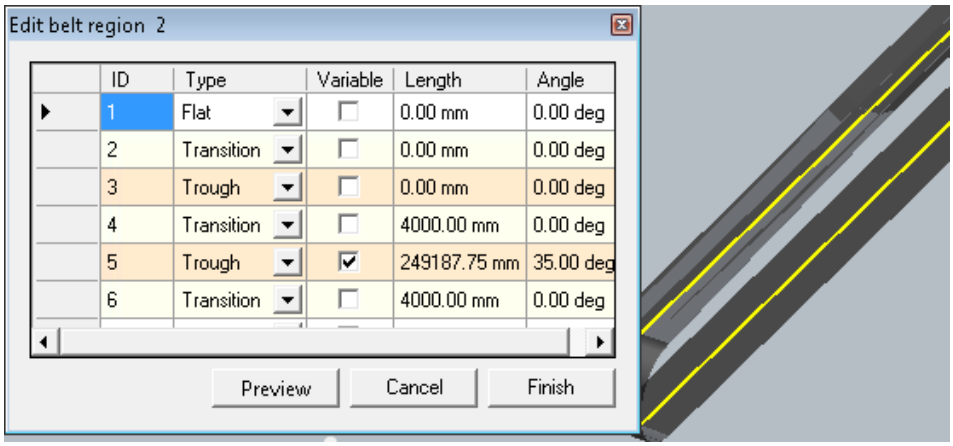
What do you want to do?

- *Define properties of a belt region* (on page 53)
- *Modify properties of a belt region* (on page 54)

Define properties of a belt region

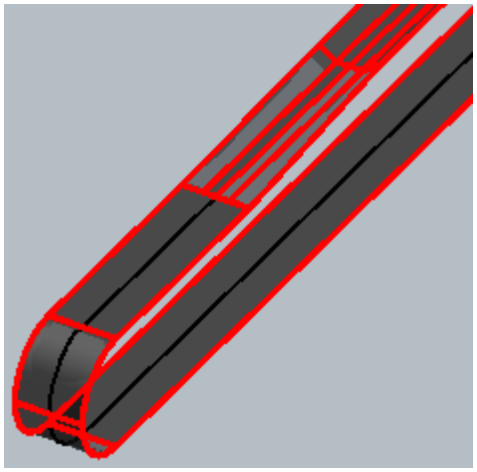
1. Click **Place Conveyor Belt** on the vertical toolbar.
2. In the graphic view, select a conveyor belt created with **Place Conveyor Belt**. Select the belt at the location of the required region.

   *The centerline of the belt highlights, and the **Edit Belt Region** dialog box displays. The region number displays in the dialog box title.*



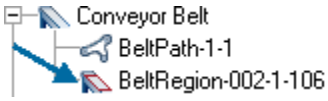
3. Change the required properties of the region. To add a zone not currently in the region, change the value of **Length** for the zone from **0** to the required value.
4. To see a preview of the changes, click **Preview**.
5. To accept the changes, click **Finish**.

   *Changes to the belt region display in the graphic view, and a belt region object displays in the **Workspace Explorer**.*

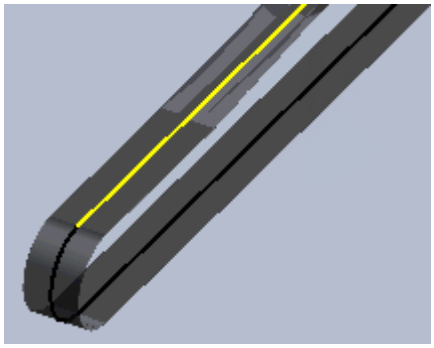


Modify properties of a belt region

1. Click **Place Conveyor Belt** on the vertical toolbar.
2. In the **Workspace Explorer**, select a belt region. Belt regions display under a conveyor belt parent:



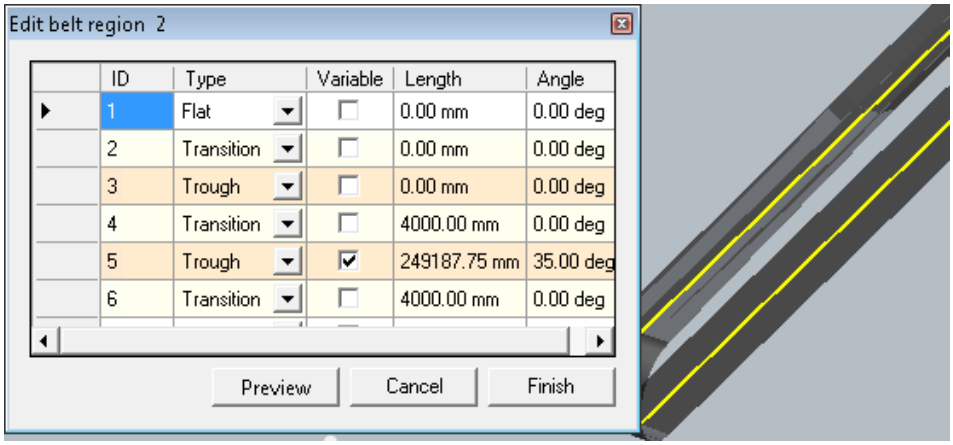
   *The centerline of the belt region highlights, and the **Edit Belt Region** dialog box displays. The region number displays in the dialog box title.*



   -OR-

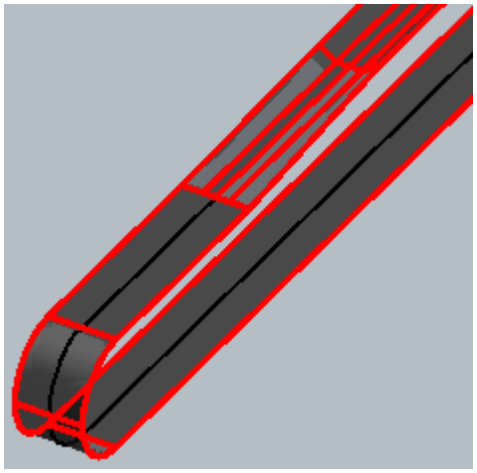
   In the graphic view, select a conveyor belt created with **Place Conveyor Belt**. Select the belt at the location of the required region.

*The centerline of the belt highlights, and the **Edit Belt Region** dialog box displays. The region number displays in the dialog box title.*



3. Change the required properties of the region. To add a zone not currently in the region, change the value of **Length** for the zone from **0** to the required value.
4. To see a preview of the changes, click **Preview**.
5. To accept the changes, click **Finish**.

Changes to the belt region display in the graphic view.



Edit Belt Region Dialog Box

Defines properties for each zone of the selected belt region. The region number displays in the dialog box title.

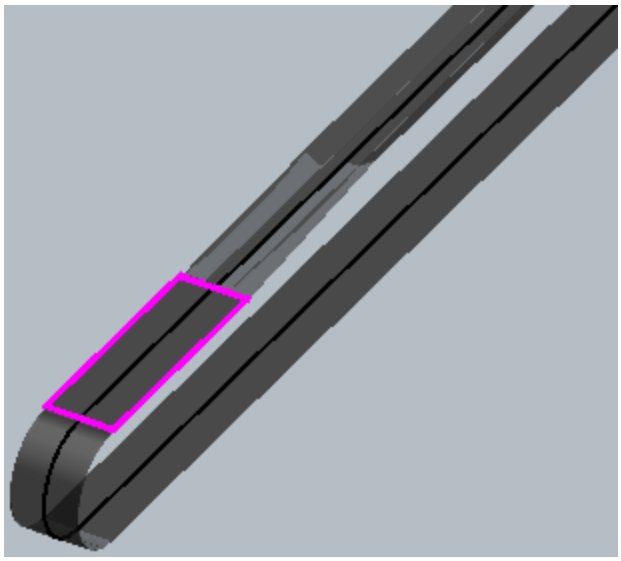
ID

Defines the zone number. Zone numbers increase in the direction of belt motion.

Type

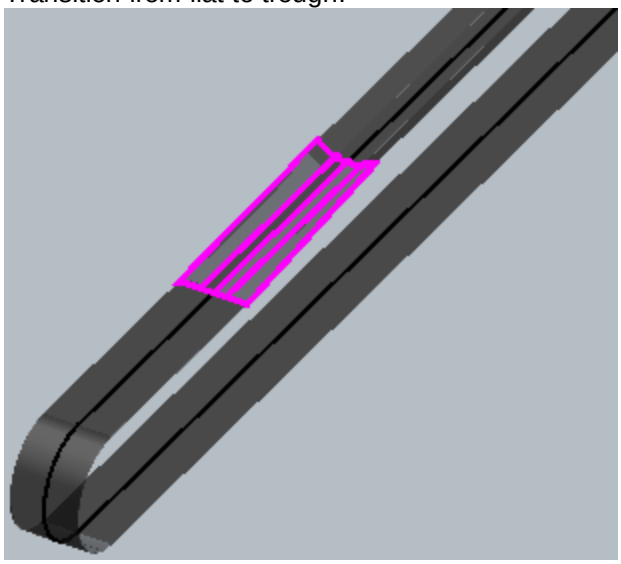
Defines the type of zone:

- **Flat** - The belt has a flat cross-section.

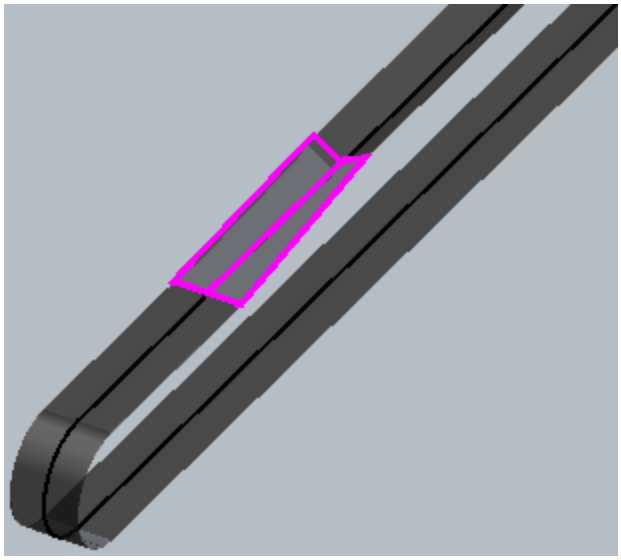


- **Transition** - The belt cross-section transitions from the preceding flat cross-section to the next trough or vee cross-section.

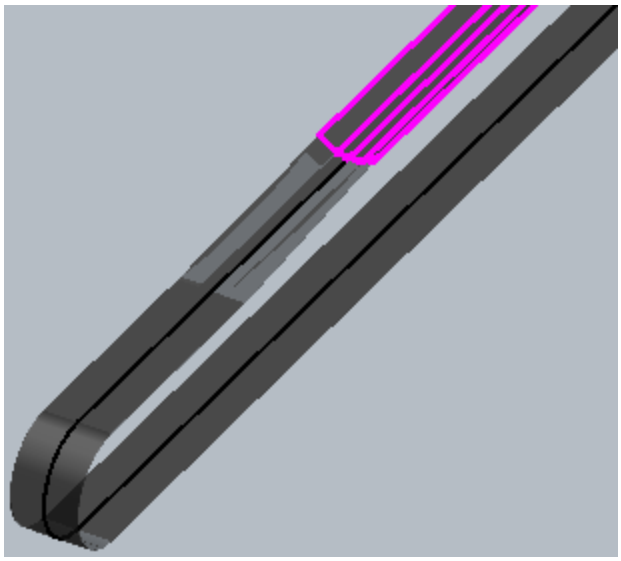
  Transition from flat to trough:



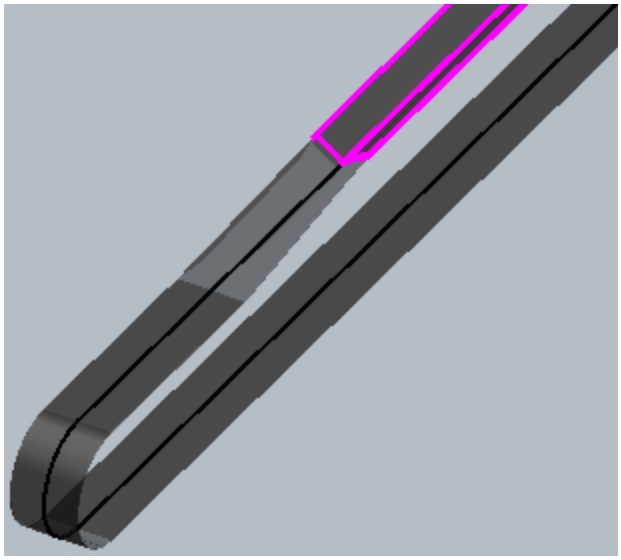
Transition from flat to vee:



- **Trough** - Rollers shape the belt cross-section into a trough with a flat bottom and sloped sides. Select **Trough** from the list.



- **Vee** - Rollers shape the belt cross-section into a V shape. Select **Vee** from the list.



Variable

Specifies that the length of the selected zone adjusts based on the length of other zones. One zone must always be selected. You can change the selected zone.

Length

Specifies the length of the zone. If you do not want to use a zone, set the value to **0**.

Angle

Specifies the angle of the sides of a **Trough** or **Vee** zone.

Preview

Displays a preview of changes made to the region in the graphic view.

Cancel

Ignores all changes and closes the dialog box.

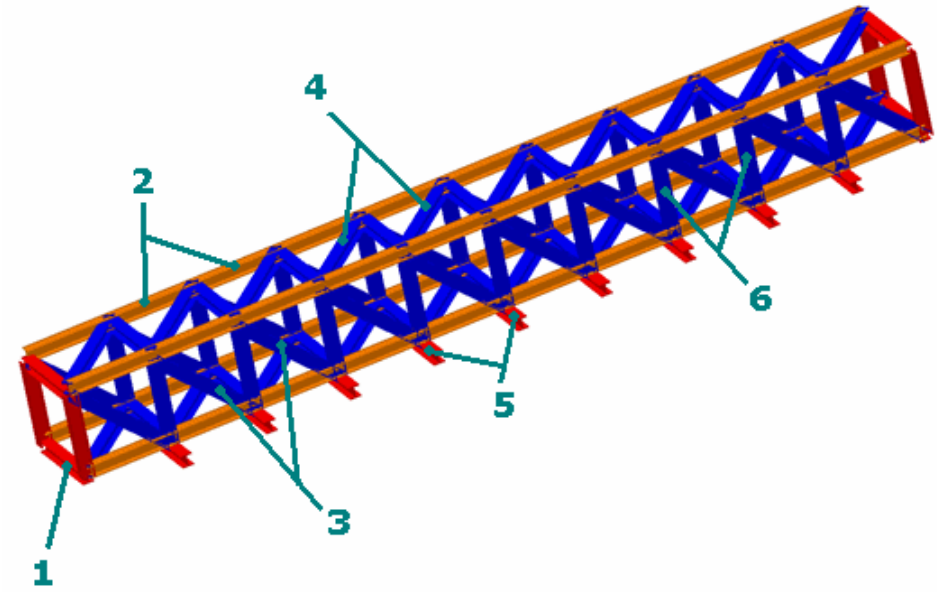
Finish

Accepts changes made to the region and closes the dialog box.

SECTION 6

Truss Wizard

Opens a wizard that steps you through the process of creating a new truss. A truss is a structural assembly used to support a conveyor belt, as well as mount equipment and parts such as idlers, walkways, and wind shields. A truss usually consists of walkway supports, four main girders, two end frames, posts, vertical bracings, and horizontal bracings. A typical truss is shown below:



1 - End frame
2 - Main girder
3 - Vertical bracing
4 - Horizontal bracing
5 - Walkway supports
6 - Posts

You can create linear and curved trusses with a variety of bracing, including the following:

- Cross
- Chevron
- N-N type
- Mirrored chevron
- Mirrored N-N type
- Cross-type vertical
- Chevron-type horizontal

Parametric Capabilities

When you create a truss, all of its structural members are parametrically associated to grids. The grids are created using the MHE grid system and can be rotated in any direction. Whenever the grids are modified, the software automatically modifies the truss members as well. The following truss parameters can be modified by the grids:

- Depth
- Width
- Spacing between posts (sets of coplanar stringers and girders)

IMPORTANT **Truss Wizard** is only used for initial creation of the truss. After the truss is created and the **Truss Wizard** dialog box is closed, truss structure is placed as members from the Structure task. Use the Structure task commands to edit the truss members after placement.

Truss Wizard Dialog Box (on page 63)

What do you want to do?

- *Create a new truss* (on page 60)
- *Change the depth of a truss* (on page 61)
- *Change the width of a truss* (on page 62)
- *Change the spacing between posts* (on page 62)

Create a new truss

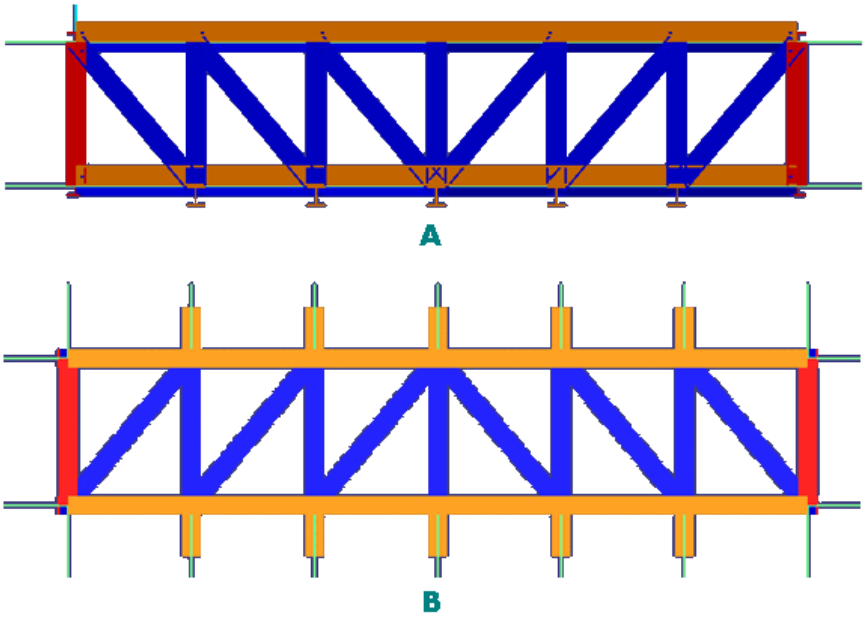
1. In the Material Handling task, click **Truss Wizard** on the vertical toolbar.

   *The **Truss Wizard** dialog box appears.*
2. Use the tabs in the **Truss Wizard** dialog box to specify the requirements for the truss. For more information, see *Truss Wizard Dialog Box* (on page 63).
3. Click **OK**.

   TIP You can create a default truss by using the default member values on the tabs.

   The software creates the truss according to the options that you have defined.

The following example shows two different views of a completed linear truss. Illustration **A** shows the elevation view of a truss with N-N type mirrored bracings. Illustration **B** shows the same truss in plan view.



NOTE To add packing between trusses and idler belt components, see *Idler Packing Thickness (Sketch 2D - Custom Command)* (on page 180).

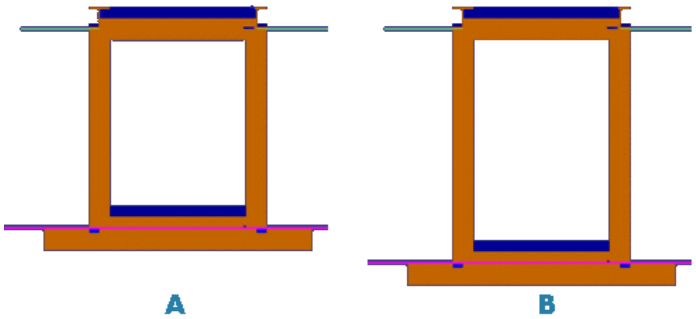
Change the depth of a truss

1. In the Grids task, click **Select** on the vertical toolbar.
2. Select **Elevation Plane** in the **Locate Filter** box.
3. Select any elevation plane in the grids system of the existing truss.

   *The **Place Grid Plane** ribbon appears.*
4. In the **Offset** box, type the required value and then press **Enter**.

   The software modifies the depth of the truss by the value specified.

The following example shows the results of modifying the depth. Illustration **A** shows a truss with a depth of -2m. Illustration **B** shows the truss after modifying the depth to -3m



Change the width of a truss

1. In the Grids task, click **Select** on the vertical toolbar.
2. Select **Grid Plane** in the **Locate Filter** box.
3. Select the first Y-axis plane in the grid system of the truss to modify.

   *The **Place Grid Plane** ribbon appears.*
4. In the **Offset** box, type the required value and then press **Enter**. Type a negative value to make the truss wider. Type a positive value to make the truss narrower.
5. Select the second Y-axis plane in the grid system of the truss to modify.

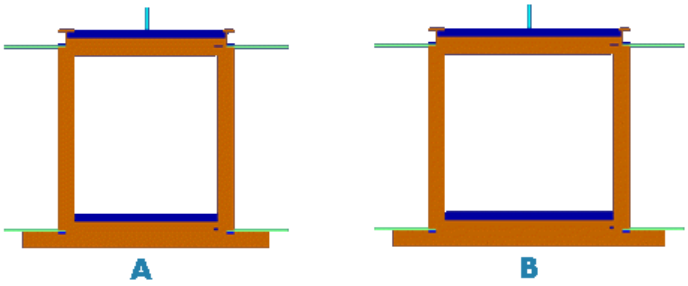
   *The **Place Grid Plane** ribbon appears.*
6. In the **Offset** box, type the required value and then press **Enter**.

   TIP Type a positive value to make the truss wider. Type a negative value to make the truss narrower.

   The software modifies the width of the truss as specified.

IMPORTANT To maintain the symmetry of the truss, you must define the same offset value for both planes, but define the values in opposite directions. Use half of the required change in width as an offset value in each direction.

The following example shows the results of modifying the width. Illustration **A** shows a truss with a width of 2 m. Illustration **B** shows the truss after modifying the width to 3 m by offsetting 0.5 m in each direction.



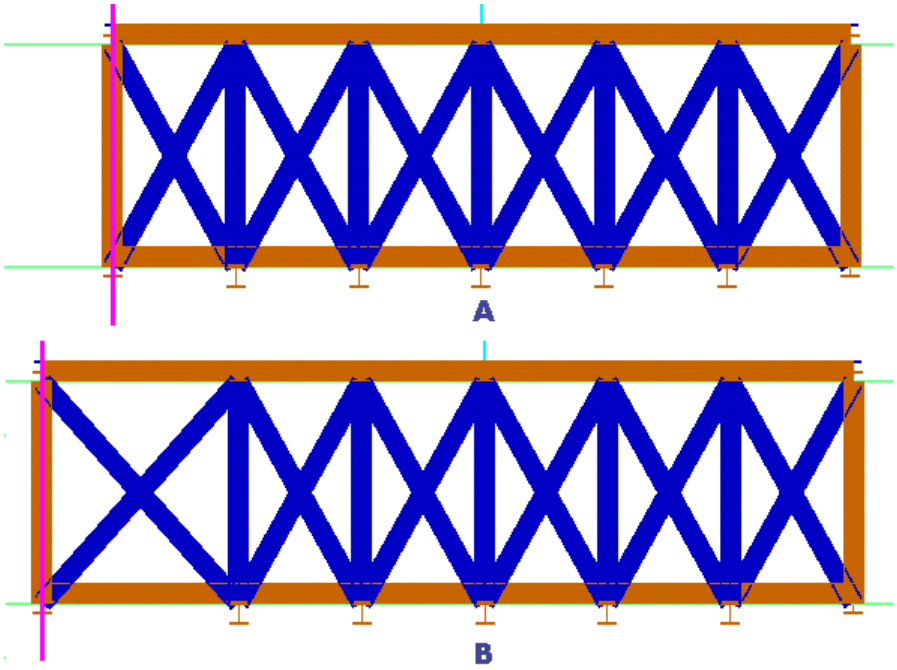
Change the spacing between posts

1. In the Grids task, click **Select** on the vertical toolbar.
2. Select **Grid Plane** in the **Locate Filter** box.
3. Select an X-axis plane in the grid system of the truss to modify.
4. Select a plane coincident with the post to move.

   *The **Place Grid Plane** ribbon appears.*
5. In the **Offset** box, type the required value and then press **Enter**.

   The software modifies the width of the truss as specified.

The following example shows the results of modifying the spacing. Illustration **A** shows the original truss. Illustration **B** shows the truss after moving the first post -6 m.



Truss Wizard Dialog Box

Specifies the properties for the truss that you are creating.

OK

Creates the truss and closes the dialog box

Apply

Creates the truss and leaves the dialog box open. After creating the truss, you can change options and click **Apply** again to place another truss in the graphic view.

Close

Closes the dialog box.

★IMPORTANT **Truss Wizard** is only used for initial creation of the truss. After the truss is created by clicking either **OK** or **Apply**, truss structure is placed as members from the Structure task.

Topics

General Tab (Truss Wizard)

System

Select the system to which the structural members of the truss are assigned. You can define new systems in the Systems and Specifications task. Select **More** to display all systems defined in the workspace or the model.

Type of Truss

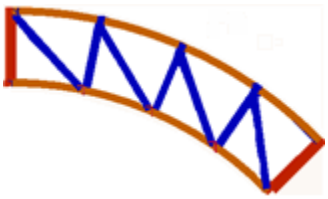
Linear Truss

Select to create a linear truss.

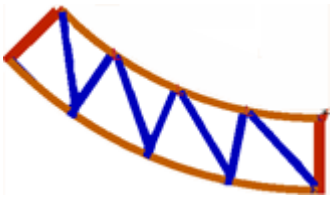
Curved Truss

Select to create a curved truss.

- **Elevation - Convex** - Defines the curved truss as convex in the elevation direction.



- **Elevation - Concave** - Defines the curved truss as concave in the elevation direction.



 **Preview**

Displays an image with the dimension standards for the selected truss type. The image displays in a separate window.

Truss Dimensions

Length

Type the overall truss length, not including offset from the placement point. For more information, see **Left WorkPoint** and **Right WorkPoint** below.

Width

Type the width dimension for the truss.

Depth

Type the depth of the truss. This specifies the distance between the upper and lower stringers.

Radius

Type the radius of the truss. This option is only available when **Curved Truss** is selected.

Copy to Catalog Name

Specify whether a copy of the truss is saved to the catalog for later reuse. Select to automatically save the new truss to the catalog in the **Conveyor System** folder, If not selected, the new truss is not copied to the catalog. When you select this option, you must type the name of the truss as it will appear in the catalog. The name must be unique across the entire catalog.

NOTE This option is available only if you have write permissions to the catalog.

Placement Point

Select the placement of the truss relative to the axis and save this information to the catalog.

Walkway Width

Define the width of the walkway. Acceptable values are in the range 0 mm to 2000 mm.

No. of Posts

Specify the number of posts required between end frames. If you specify an even number, the **Mirror** option on the **Bracings** tab is unavailable.

Grids Data

Display Grid Lines

Specify whether the created grid lines are to be displayed in the graphic window.

Specify the model *global* coordinate system coordinates used to locate the origin of the truss *local* coordinate system:

- **East(X)** - Type the eastern or X coordinate.
- **North(Y)** - Type the northern or Y coordinate.
- **Up(Z)** - Type the elevation or Z coordinate.

Coordinate System Name

Define the name of the coordinate system that you are creating for the truss. For a linear truss, select a rectangular coordinate system. For a curved truss, select a radial coordinate system. For more information, see *Grid Wizard* in the *Grids User's Guide*.

Left WorkPoint

Type the distance of the placement control point from the left end of the truss.

Right WorkPoint

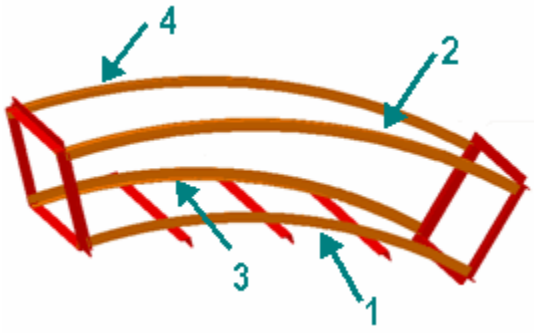
Type the distance of the placement control point from the right end of the truss.

Stringers and Walkway Tab (Truss Wizard)

Specifies all truss structural members except end frames.

Stringers

A truss has four stringers:



Member Type

Select the type of member that you want to place, such as **Stringer**, **Beam**, **Joist**, **Girder**, or **Column**.

Standard

Displays the corresponding steel design standard. This value is defined in the catalog and is read-only.

Section

Define the cross-section for the member. If you know the section name that you want, type it in. Use the asterisk [*] character wild card to see all sections containing that text. For example, type W10X* to see all W10X sections in the catalog. Select **More** to browse the catalog.

Material

Select the material type for the structural member.

Grade

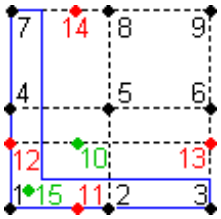
Select the material grade for the structural member.

Preview

Displays an image with the stringer locations. The image displays in a separate window.

Cardinal Point

Specify the relative position of the structural section to the member placement line. Fifteen cardinal positions are available. The location of cardinal points 10 (center-of- gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. A value is defined for each stringer of the truss, **Stringer1, Stringer2**, **Stringer3**, and **Stringer4**.

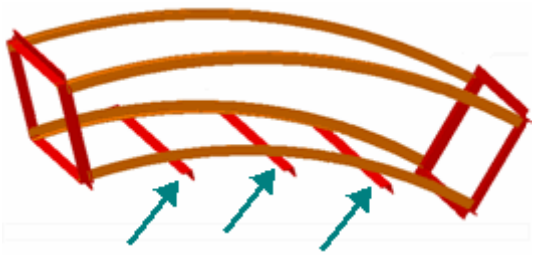


Angle

Specify the orientation of the stringer cross-section. A value is defined for each stringer of the truss, **Stringer1, Stringer2**, **Stringer3**, and **Stringer4**.

Walkway Support

A truss typically has multiple walkway supports:



Member Type

Select the type of member that you want to place, such as **Support**, **Landing**, **Stair Attachment**, **Rung**, or **Rail**.

Standard

Displays the corresponding steel design standard. This value is defined in the catalog and is read-only.

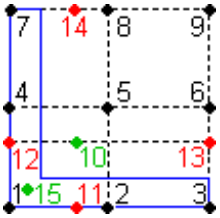
Section

Define the cross-section for the member. If you know the section name that you want, type it in. Use the asterisk [*] character wild card to see all sections containing that text. For example, type W10X* to see all W10X sections in the catalog. Select **More** to browse the catalog.

Cardinal Point

Specify the relative position of the structural section to the member placement line. Fifteen cardinal positions are available. The location of cardinal points 10 (center-of- gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the

member and the center-of-gravity point of the section define cardinal points 12 and 13.



Material

Select a material for the member. Materials are defined in the **AllCommon.xls** workbook or in the Catalog task.

Grade

Select a material grade for the member. Material grades are defined in the **AllCommon.xls** workbook or in the Catalog task.

Angle

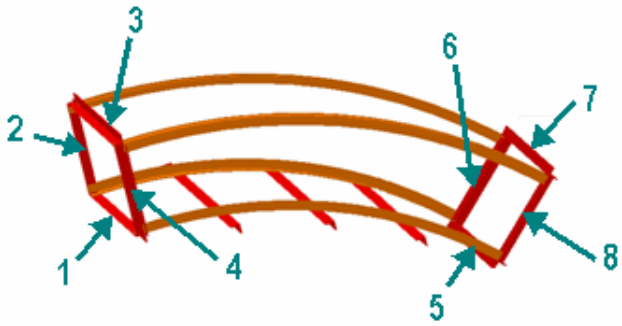
Specify the orientation of the support cross-section.

Preview

Displays an image with the walkway support locations. The image displays in a separate window.

End Frames Tab (Truss Wizard)

Specifies end frame structural members. A truss has eight end frame members:



Member Type

Select the type of member that you want to place, such as **Column**, **Beam**, or **Girder.**

Standard

Displays the corresponding steel design standard. This value is defined in the catalog and is read-only.

Section

Define the cross-section for the member. If you know the section name that you want, type it in. Use the asterisk [*] character wild card to see all sections containing that text. For example, type W10X* to see all W10X sections in the catalog. Select **More** to browse the catalog.

Material

Select the material type for the structural member.

Grade

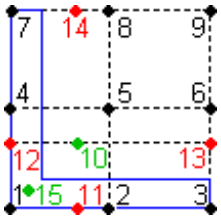
Select the material grade for the structural member.

Preview

Displays an image with the end frame member locations. The image displays in a separate window.

Cardinal Point

Specify the relative position of the structural section to the member placement line. Fifteen cardinal positions are available. The location of cardinal points 10 (center-of- gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. A value is defined for each end frame member of the truss, **EndFrame1** through **EndFrame8**.



Angle

Specify the orientation of the end frame member cross-section. A value is defined for each end frame member of the truss, **EndFrame1** through **EndFrame8**.

Bracing Tab (Truss Wizard)

Mirror

Specify whether the truss is mirrored. If you select this option, the software places up to half of the truss and then mirrors that portion to complete placement. This option is not available if you type an even number in the **No. of Posts** box on the **General** tab.

The following options are available for both **Horizontal Bracing** (on the horizontal planes of the truss) and **Vertical Bracing** (on the vertical planes of the truss):

Bracing Type

Specify the type of bracing to place between each pair of stringers (horizontal bracing) or girders (vertical bracing). Select **Cross** to place two braces in an X configuration. Select **Chevron** to place one diagonal brace with each brace going end-to-end along the truss to form V-shapes. Select **N-Type** to place one diagonal brace with each brace forming an N-shape along the truss.

Member Type

Select the type of cross bracing member that you want to place, such as a brace or vertical brace.

Preview

Displays an image with the bracing member locations. The image displays in a separate window.

Standard

Displays the corresponding steel design standard. This value is defined in the catalog and is read-only.

Section

Define the cross-section for the bracing members. If you know the section name that you want, type it in. Use the asterisk [*] character wild card to see all sections containing that text. For example, type W10X* to see all W10X sections in the catalog. Select **More** to browse the catalog for the section to use. Sections are defined in the reference data.

Material

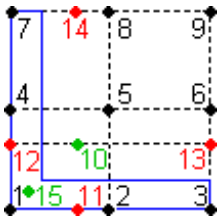
Select the material for the bracing member.

Grade

Select the material grade for the bracing member.

Cardinal Point

Specify the relative position of the structural section to the cross bracing member placement line. Fifteen cardinal positions are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. A value is defined for each horizontal and vertical bracing member of the truss.



Angle

Specify the orientation of the end frame member cross-section. A value is defined for each horizontal and vertical bracing member of the truss.

SECTION 7

Place Belt Components

Places belt components such as equipment, modules, and miscellaneous objects (control points) along the path of a belt profile in Sketch 2D.

If you select a belt profile that was created by **Place Conveyor Belt** (on page 23), Smart 3D creates a parametric relationship between the belt profile and the equipment and modules. Miscellaneous objects do not maintain a relationship with the belt. For example, if you move the belt profile, the belt components also move to maintain their relationship to the belt profile.

Place Belt Components Ribbon

Place Belt Component

Activates the place belt component command.

Review Symbol Files

Allows you to review symbol files for belt profiles. Select the belt profile within the Workspace Explorer to turn on **Review**. For more information, see *Review Dialog Box* (on page 77).

Select Belt Profile

Select the appropriate belt profile to place the components.

Sketch2D Control

Displays options for the Sketch 2D environment where you can work with the belt components using **Place Belt Components**. For more information, see *2D Automation in Sketch 2D* (on page 159).

Finish

Places the belt components.

Update 3D

Updates the belt components in 3D environment with changes made in Sketch 2D environment.

Select Parent

Select the parent system into which the belt components will be placed.

Placement Mode

Select the appropriate method for handling belt components.

- **Add, Modify** – Adds new objects in the model. Modifies existing objects in the model, but does not delete objects in the model.
- **Add, Modify, Delete** – Adds new objects in the model. Modifies existing objects in the model, and deletes objects in the model.

Symbol File Name

Type a name for the symbol file.

Type of Belt Components

Select the appropriate component type. For example, select **Equipment** to place equipment type components such as idlers. You can place all components (equipment, modules and miscellaneous) by selecting the **All** option.

2D Sketcher Options

NOTE Some 2D Sketcher options listed below may not be applicable for the active command.

Sketching Plane

Specifies the sketching plane. This is the first step in defining the object.

Add Intersecting Item

Allows you to select objects that intersect the sketching plane in the 3D environment. You see the selected objects in the Sketch 2D environment when you are drawing.

Add Projection Item

Allows you to select objects that do not intersect the sketching plane in the 3D environment. The objects are projected onto the sketching plane, and you see the selected objects in the Sketch 2D environment when you are drawing.

Sketch 2D

Opens the Sketch 2D environment in which you can draw.

Coincident Plane

Specifies that you want to sketch on the plane that you select.

Offset from Plane

Specifies a sketching plane that is offset from a plane that you select. If you choose this option, you must define the offset distance.

Angle from plane

Specifies a sketching plane that is at a specified angle from a plane that you select. If you choose this option, you must define an axis of rotation and the angle or slope.

Plane by Point and Vector

Specifies the sketching plane using two points to define a vector normal to the sketching plane and a third point to define the sketching plane position along the vector.

Plane by Three Points

Specifies the sketching plane using three points that you specify in the model.

Plane by Vectors Normal

Specifies the sketching plane as being normal to another plane that you select and having a rotation parallel to a vector that you define.

Offset lock

Locks the offset value, disabling updates of the offset value by mouse moves. This option is only available when you use the **Offset from a Plane** option.

Offset

Specify the offset distance for the sketching plane from the selected plane. You can specify the offset dynamically in graphics or by typing the distance. This option is only available when you use the **Offset from a Plane** option.

Angle

Specify the angle at which to place the sketching plane relative to the selected plane. You have to define the axis of rotation using two points before you can define the angle. This option is only available when you use the **Angle from plane** option.

Step

Specifies the angle step. The step is incremented by this value when the cursor is moved in the graphic view. This option is only available when you use the **Angle from plane** option.

Select Vector

Select the vector normal to the sketching plane. This option is only available when you use the **Plane by Point and Vector** option.

Define Point

Specify the point along the vector at which to place the sketching plane. This option is only available when you use the **Plane by Point and Vector** option.

Define Point 1

Specify the location of the first of three points that will define the sketching plane. This option is only available when you are using the **Plane by Three Points** option.

Define Point 2

Specify the location of the second of three points that will define the sketching plane. This option is only available when you are using the **Plane by Three Points** option.

Define Point 3

Specify the location of the third of three points that will define the sketching plane. This option is only available when you are using the **Plane by Three Points** option.

Review Dialog Box (on page 77)

What do you want to do?

- *Create belt components* (on page 74)
- *Modify a belt component in Sketch 2D* (on page 75)
- *Modify a belt component in 3D* (on page 76)
- *Modify a belt object's relationship to a belt* (on page 118)
- *Delete a symbol file* (on page 76)

Create belt components

CAUTION Do not delete or modify the **Note Text** that displays the position and orientation for module properties.

1. In the Material Handling task, click **Place Belt Components** on the vertical toolbar.

   The ***Place Belt Components*** ribbon appears.

2. Click **Place Belt Component**.

   Select Belt Profile *is the active option.*

3. Select a belt profile, either from the **Workspace Explorer** or from the graphic view.

4. Click **Sketch 2D Control**.

   The sketch plane of the belt profile is automatically selected.

5. Use **Add Intersecting Item** and **Add Projection Item** as needed to add items to the sketch.

6. Click **Sketch 2D**.

   The Sketch 2D environment opens. The belt profile, intersection items, and projection items are displayed based on the sketching plane you have selected.

7. Click **Place Belt Components**.

   *The **Place Belt Components** dialog box appears.*

   NOTE If **Place Belt Components** is not available, see *Add custom commands to the Sketch 2D toolbar* (on page 161).

8. On the **Configuration** tab, select an appropriate component type for **Select Belt Component**.

9. Browse to the folder containing the 2D symbols and associated command configuration (.LST) files.

   NOTE This folder is usually a subfolder of the SharedContent symbols share.

10. Select the **Place** tab:

    - If you selected **Equipment** in **Select Belt Component,** see *Place equipment symbols in 2D* (on page 162).
    - If you selected **Modules** in **Select Belt Component,** see *Place module symbols in 2D* (on page 163).
    - If you selected **Misc** in **Select Belt Component,** see *Place miscellaneous symbols in 2D* (on page 164).

11. After placing the components, click **Finish.**

    The Sketch 2D environment closes.

12. In the 3D environment, select a system in **Select Parent**.

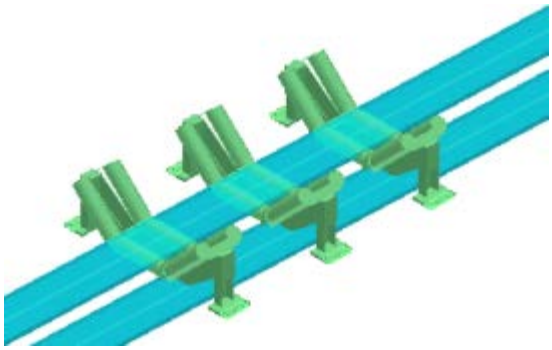
13. For **Placement Mode,** select **Add, Modify.**

14. For **Symbol File Name,** type a name.

15. For **Type of Belt Components,** select **All**.
16. Select **Update 3D** to view the newly-created belt components.

    ★IMPORTANT If you do not select this option, the components are not created in the 3D environment.
17. Click **Finish**.

    The equipment symbols are added to the model. Idlers are shown in the example below.



NOTE To add packing between idlers and trusses, see *Idler Packing Thickness (Sketch 2D - Custom Command)* (on page 180).

Modify a belt component in Sketch 2D

In this workflow, the belt profile and sketching plane are retrieved from the symbol file so that you can modify the belt component.

1. In the Material Handling task, click **Place Belt Components** on the vertical toolbar.

   *The **Place Belt Components** ribbon appears.*
2. Click **Modify or Delete Belt Components**.

   The ***Review*** dialog box opens.
3. In **Select Mode**, select **Modification**.
4. Select the needed symbol file from the list.
5. Click **OK**.

   The ***Review*** dialog box closes.
6. On the ribbon bar, click **Sketch2D**.

   TIP You can also add intersection and projection items.

   A message box appears stating "Rad 2D file is updated according to the 3D changes."
7. Click **OK**.

   The Sketch 2D environment opens.

8. Modify the 2D symbol. To place new component(s), see *Create belt components* (on page 74).

   TIP You can also use **QuickPick** to select 2D symbols and move them to the required location. When you pause the mouse in the graphic view in an area where there are multiple objects, the cursor changes to the **QuickPick** question mark. A single left-click enables the **QuickPick** feature and displays the **QuickPick** dialog box in a semitransparent mode.

9. Click **Finish.**

   You are returned to the 3D environment.

10. In **Select Parent,** select a system.
11. In **Placement Mode**, select:
    - **Add, Modify** to modify the component.
    - **Add, Modify, Delete** to delete the component.
12. In **Type of Belt Components,** select **All**.
13. Click **Finish**.

Modify a belt component in 3D

After placing belt components, you can change orientation, location and delete any component(s) in the 3D environment. The 3D to 2D parametric capability synchronizes 3D graphics with 2D symbols. Therefore, the changes made to the placed components in the 3D environment are reflected in the 2D environment. Follow the below steps to achieve this functionality:

1. In the 3D environment, modify any placed belt component(s).
2. Click **Place Belt Components**  .
3. Click **Modify or Delete Belt Components**  .

   *The **Review** dialog box opens.*

4. Select the symbol file of the modified belt components.
5. Click **Sketch2D**  .

   A message box appears stating "Rad 2D file is updated according to the 3D changes."

6. Click **OK**.

   The Sketch 2D environment opens. 2D symbols are updated with changes made in the 3D environment.

Delete a symbol file

1. Click **Review**  .

   *The **Review** dialog box opens.*

2. In **Select Mode,** select **Deletion**.
3. Select the needed symbol file from the list.

4. Click **OK**.

   The symbol file is deleted from the database and removed from the symbol list.

5. Click **Close**.

Review Dialog Box

Select Mode

Select **Modification** to modify the symbol file. Select **Deletion** to delete the symbol file.

S.No, Belt Name, **Symbol Name, Date Created**

A list of all symbol files created with the **Place Belt Components** command. Select the needed symbol file.

OK

Accepts the selection.

Close

Closes the dialog box and resets the **Place Belt Components** ribbon bar to its default values.

SECTION 8

Place Trestle

Places a trestle on the conveyor in Sketch 2D. Smart 3D creates a parametric relationship between the belt profile and the trestle. For example, if you move the belt profile, the trestle also moves to maintain its relationship to the belt profile.

NOTE Before placing a trestle on the conveyor, you must use *Place Belt Components* (on page 71) to create a symbol file for the trestle.

Place Trestle Ribbon

Controls parameters for placing a trestle in the model.

Review Symbol Files

Allows you to review symbol files for belt profiles. Select the belt profile within the Workspace Explorer to turn on **Review Symbol Files**. For more information, see *Review Dialog Box* (on page 77).

Sketch2D

Displays options for the Sketch 2D environment where you can work with the trestle. For more information, see *2D Automation in Sketch 2D* (on page 159).

Finish

Places the trestle.

Select Parent

Selects the parent system for the trestle. Click **More** to display the common **Select System** dialog box.

Placement Mode

Selects the method for handling imported equipment.

- **Add, Modify** - Adds new objects on the export list. Modifies existing objects on the export list. Does not delete objects in the model.
- **Add, Modify, Delete** - Adds new objects on the export list. Modifies existing objects on the export list. Deletes objects in the model but not on the export list.

2D Sketcher Options

NOTE Some 2D Sketcher options listed below may not be applicable for the active command.

Sketching Plane

Specifies the sketching plane. This is the first step in defining the object.

Add Intersecting Item

Allows you to select objects that intersect the sketching plane in the 3D environment. You

see the selected objects in the Sketch 2D environment when you are drawing.

Add Projection Item

Allows you to select objects that do not intersect the sketching plane in the 3D environment. The objects are projected onto the sketching plane, and you see the selected objects in the Sketch 2D environment when you are drawing.

Sketch 2D

Opens the Sketch 2D environment in which you can draw.

Coincident Plane

Specifies that you want to sketch on the plane that you select.

Offset from Plane

Specifies a sketching plane that is offset from a plane that you select. If you choose this option, you must define the offset distance.

Angle from plane

Specifies a sketching plane that is at a specified angle from a plane that you select. If you choose this option, you must define an axis of rotation and the angle or slope.

Plane by Point and Vector

Specifies the sketching plane using two points to define a vector normal to the sketching plane and a third point to define the sketching plane position along the vector.

Plane by Three Points

Specifies the sketching plane using three points that you specify in the model.

Plane by Vectors Normal

Specifies the sketching plane as being normal to another plane that you select and having a rotation parallel to a vector that you define.

What do you want to do?

- *Place a trestle* (on page 79)
- *Modify a trestle* (on page 80)
- *Delete a trestle* (on page 81)

Place a trestle

1. Click **Place Trestle** on the vertical toolbar.

   *The **Place Trestle** ribbon appears.*

2. Click **Select Symbol File**.

   *The **Review** dialog box appears.*

3. Select the symbol file for the required trestle, and then click **OK**.
4. Select the parent system for the trestle in the **Select Parent** list.

   TIP If needed, click **More** to display the **Select System** dialog box, and then select the required system.
5. Specify the symbol file name in the **Symbol File Name** box.
6. Click **Add Projection Item** .
7. Select the items to add. You can select items in the model or in the **Workspace Explorer**. These items display in the Sketch 2D environment.

   The selected items highlight.
8. Click **Sketch2D** .

   The Sketch 2D environment displays.
9. Fit the view so that all of your projected items display.
10. Add the **MHPlace2DTrestle.ocx** custom command to the toolbar if it does not already display.
11. Click **Place 2D Trestle** .

    *The **Place 2D Trestles** dialog box displays.*
12. Set up the parameters as required. For more information, see *Place 2D Trestles Dialog Box* (on page 176).

    The software prompts you to pick the first point.
13. Select the top point for the trestle.

    The software prompts you to pick the second point.
14. Select the bottom point for the trestle.
15. Continue to define the parameters and points for the trestle parts.
16. Click **Finish** in the Sketch 2D environment.

    The software returns to the model, and highlights the trestles.
17. Click **Finish** on the **Place Trestle** ribbon.

    The software adds the trestles to the model.

Modify a trestle

1. Click **Place Trestle** on the vertical toolbar.

   *The **Place Trestle** ribbon appears.*
2. Click **Select Symbol File** .

   *The **Review** dialog box appears.*
3. Select the symbol file for the trestle to modify, and then click **OK**.

4. Click **Sketch2D** .

   The Sketch 2D environment displays.

5. Fit the view so that all of your projected items display.
6. Add the **MHPlace2DTrestle.ocx** custom command to the toolbar if it does not already display.
7. Click **Place 2D Trestle** .

   *The **Place 2D Trestles** dialog box displays.*

8. Set up the parameters as required. For more information, see *Place 2D Trestles Dialog Box* (on page 176).

   The software prompts you to pick the first point.

9. Select the top point for the trestle.

   The software prompts you to pick the second point.

10. Select the bottom point for the trestle.
11. Continue to define the parameters and points for the trestle parts.
12. Click **Finish** in the Sketch 2D environment.

    The software returns to the model, and highlights the trestles.

13. Click **Finish** on the **Place Trestle** ribbon.

    The software updates the trestle in the model.

Delete a trestle

1. Click **Place Trestle** on the vertical toolbar.

   *The **Place Trestle** ribbon appears.*

2. Click **Select Symbol File** .

   *The **Review** dialog box appears.*

3. Select the symbol file of the trestle to delete.
4. If necessary, set **Select Mode** to **Add, Modify, Delete**.
5. Click **OK**.
6. Click **Sketch2D** .

   The Sketch 2D environment displays.

7. Delete the trestles.
8. Click **Finish** in the Sketch 2D environment.

   The software returns to the model, and highlights the trestles.

9. Click **Finish** on the **Place Trestle** ribbon.

   The software updates the trestle in the model.

SECTION 9

Place Chute Shapes

Places chute shapes in the model.

NOTE You can also use chute shape geometric construction macros for quick layout of chute design. Download the macros from *Intergraph Smart Support* (*https://smartsupport.intergraph.com*) and install according to the provided instructions.

Place Chute Shapes Ribbon

Controls parameters for placing chute shapes in the model.

Place Chute Shape

Places the chute shape.

Modify/Delete Chute Shapes

Displays the **Review** dialog box. For more information, see *Review Dialog Box* (on page 77).

Select Belt Profile

Selects the elevation and side elevation belt profiles the chutes support.

Sketch 2D for elevation layout

Selects the belt profile of the conveyer system, and enables the Sketch 2D control for elevation layout.

Sketch 2D for side elevation layout

Selects the side belt profile of the conveyer system, and enables the Sketch 2D control for side elevation layout.

Sketch2D

Displays options for the Sketch 2D environment where you can work with the chute shape. For more information, see *2D Automation in Sketch 2D* (on page 159).

Finish

Places the chute shapes.

Select Parent

Selects the parent system for the chute shape. Click **More** to display the common **Select System** dialog box.

Placement Mode

Selects the method for handling chute shapes.

- **Add, Modify** - Adds chute shapes into the 3D model, modifies existing chute shapes in the 3D model, and does not delete chute shapes in the model if they are deleted in the Sketch 2D.

- **Add, Modify, Delete** - Adds chute shapes into the 3D model, modifies existing chute shapes in the 3D model and deletes chute shapes in the model if they are deleted in the Sketch 2D.

Symbol File Name

Specifies the name of the symbol file.

Sketch 2D Options

Sketching Plane

Specifies the sketching plane. This is the first step in defining the trestle.

Add Intersecting Item

Allows you to select objects that intersect the sketching plane in the 3D environment. You see the selected objects in the Sketch 2D environment when you are drawing.

Add Projection Item

Allows you to select objects that do not intersect the sketching plane in the 3D environment. The objects are projected onto the sketching plane, and you see the selected objects in the Sketch 2D environment when you are drawing.

Sketch 2D

Opens the Sketch 2D environment in which you can draw.

Coincident Plane

Specifies that you want to sketch on the plane that you select.

Offset from Plane

Specifies a sketching plane that is offset from a plane that you select. If you choose this option, you must define the offset distance.

Angle from plane

Specifies a sketching plane that is at a specified angle from a plane that you select. If you choose this option, you must define an axis of rotation and the angle or slope.

Plane by Point and Vector

Specifies the sketching plane using two points to define a vector normal to the sketching plane and a third point to define the sketching plane position along the vector.

Plane by Three Points

Specifies the sketching plane using three points that you specify in the model.

Plane by Vectors Normal

Specifies the sketching plane as being normal to another plane that you select and having a rotation parallel to a vector that you define.

What do you want to do?

Place a chute shape

1. Click **Place Chute Shapes** on the vertical toolbar.

   *The **Place Chute Shapes** ribbon displays with the **Place Chute Shape** and **Select belt profiles** command selected.*

2. Select the belt profiles in the model.

   The belt profiles highlight.

3. Click **Sketch 2D for elevation layout** .

   *In the **Workspace Explorer**, L 0.000m, the origin plane on the Y-axis of the Belt Elevation system is selected.*

   TIP Use **Add Intersecting Item** and **Add Projection Item** as needed to add items to the sketch.

4. Click **Sketch 2D**.

   The Sketch 2D environment displays.

   TIP Fit the view so that all the selected items display.

5. Add the **MHPlace2DShapes.ocx** custom command to the toolbar if it does not already display.

   Add custom commands to the Sketch 2D toolbar (on page 161)

6. Click **Place 2D chute shapes** .

   *The **Place 2D chute shapes** dialog box displays.*

7. Select a 2D shape from the **Select 2D Symbol** list, and then place on the belt profile as required. Continue to place shapes as required.

8. Set the remaining parameters, and then click **Close**. For more information, see *Place 2D Chute Shapes Dialog Box* (on page 174).

9. Click **Finish**.

   The software returns to the 3D environment.

10. On the **Place Chute Shapes** ribbon bar, click **Sketch 2D for side elevation layout**.

    *In the **Workspace Explorer**, L 0.000m, the origin plane on the Y-axis of the Side Belt Elevation system is selected.*

11. Click **Sketch 2D** .

    The Sketch 2D environment displays.

12. Repeat steps 6 and 7.
13. Click **Close**.
14. Select the parent system for the chute shape from the **Select Parent** list.

    TIP If necessary, click **More** to display the **Select System** dialog box, and then select the system.

15. Set the **Placement mode** to **Add,Modify**.
16. Click **Finish**.

    The software adds the chute shapes to the model.

Modify a chute shape

1. Click **Place Chute Shapes** on the vertical toolbar.

   *The **Place Chute Shapes** ribbon displays with the **Place Chute Shape** command selected.*

2. Click **Modify/Delete Chute Shapes** .

   *The **Review** dialog box displays.*

3. In **Select Mode**, select **Modification**.
4. Select the needed symbol file from the list.
5. Click **OK**.

   *The **Sketch 2D for elevation layout** is selected, which means you are about to modify Belt Elevation system.*

6. On the ribbon bar, click **Sketch 2D** .

   TIP You can also add intersection and projection items.

   A message box appears stating "Rad 2D file is updated according to the 3D changes."

7. Click **OK** to close the message box.

   The Sketch 2D environment displays.

8. Click **Symbol Explorer** to display the **Attribute Viewer**.
9. Modify the 2D symbol as needed. To place a new chute shape, see *Place a chute shape* (on page 84).
10. To modify the chute shape attributes, click **Preview** .

    *The shape preview displays with the attributes you can edit. The attributes display in the **Attribute Viewer**.*

11. Change the chute shape attributes in the **Attribute Viewer** as needed.
12. Click **Finish**.

    The software returns to the 3D environment.

13. Click **Sketch 2D for side elevation layout** , to modify the side belt elevation system.
14. Repeat steps 6 through 11 as necessary.
15. Set the **Placement mode** to **Add,Modify**.
16. Click **Finish**.

    The software updates the chute in the model.

Delete a chute shape

1. Click **Place Chute Shapes** on the vertical toolbar.

   *The **Place Chute Shapes** ribbon displays with the **Place Chute Shape** command selected.*

2. Click **Modify/Delete Chute Shapes** .

   *The **Review** dialog box displays.*

3. In **Select Mode**, select **Modification**.
4. Select the needed symbol file from the list.
5. Click **OK**.

   *The **Sketch 2D for elevation layout** is selected, which means you are about to modify Belt Elevation system.*

6. On the ribbon bar, click **Sketch 2D** .

   A message box appears stating "Rad 2D file is updated according to the 3D changes."

7. Click **OK** to close the message box.

   The Sketch 2D environment displays.

8. Select the 2D symbol to delete.
9. On the **Change** ribbon bar, click **Delete** .
10. Click **Finish**.

    The software returns to the 3D environment.

11. Click **Sketch 2D for side elevation layout** , to modify the side belt elevation system.
12. Repeat steps 6 and 10.
13. Set the **Placement mode** to **Add,Modify,Delete**.
14. Click **Finish**.

    The software updates the chute in the model.

Add a trajectory path

1. Set your model view to the **Front** orientation.
2. Click **Place Chute Shapes** .

   *The **Place Chute Shapes** ribbon displays.*
3. Select the conveyer belt.
4. Select **More** from the **Select Parent** list.

   *The **Select System** dialog box displays.*
5. Select the parent for the chute.
6. Click **Sketch 2D** .

   ***Sketch 2D** opens with the selected conveyer displayed.*
7. Zoom in on the end of the conveyer to which to add the trajectory path.
8. Load the MHCreateTrajectory.ocx custom command. For more information, see *Add custom commands to the Sketch 2D toolbar* (on page 161) and *Create Trajectory (Sketch 2D - Custom Command)* (on page 175).

   TIP The MHCreateTrajectory.ocx file is delivered by default to the *[Product Folder]*\MaterialsHandling\Client\Bin\ folder.

   *The **Create Trajectory** dialog box displays.*
9. Set the parameters to meet your requirements. For more information, see *Create Trajectory Dialog Box* (on page 175).
10. Select the center of the pulley.

    The software places the trajectory path. The trajectory path displays only in the Sketch 2D environment. This aids placing head section and chute shapes.
11. Click **Finish**.

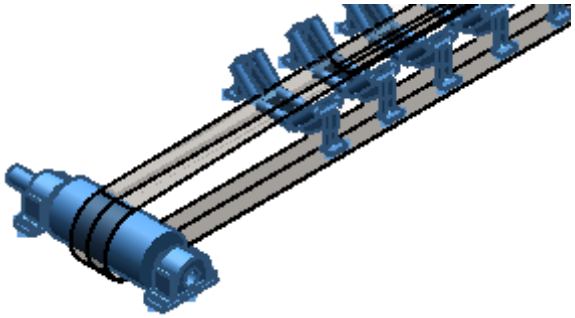
    *The software closes **Sketch 2D.***

SECTION 10

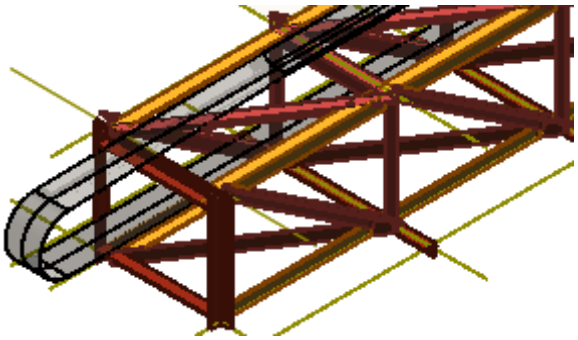
Place Belt Components in 3D

Places conveyor belt components directly in the 3D view. You can place:

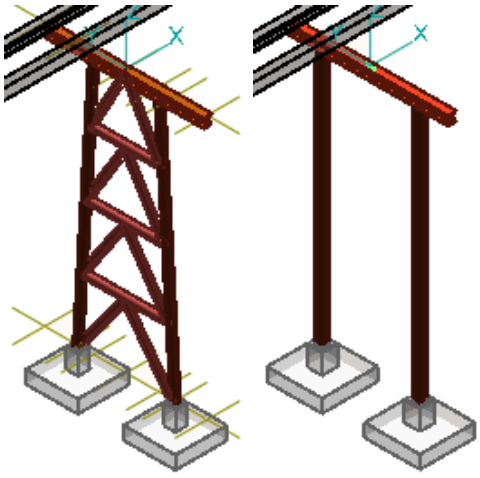
- Equipment, such as idlers and pulleys.



- Modules (such as trusses) supporting the equipment.



- Trestles and posts supporting the trusses.



Smart 3D creates a parametric relationship between the belt profile and the belt components. For example, if you move the conveyor belt, pulleys and idlers also move to maintain their relationships to the belt.

Place Belt Components in 3D Ribbon

 **Select Conveyor Belt**

Specifies the belt to associate with the components.

Finish

Saves the parent specified in the **Select Parent** box and closes the command.

Select Parent

Specifies the parent system for the belt components. The software displays the model root system as the default value. To select another system, click **More** to display the common **Select System** dialog box.

Type of Components

Defines the type of belt component to place.

- **Equipment** - Belt equipment such as idlers and pulleys. Displays the *Place Equipment dialog box* (on page 95).
- **Modules** - Belt modules such as truss structure to support the belt equipment. Displays the *Place Module dialog box* (on page 100).
- **Trestles** - Trestle structure to support the trusses. Displays the *Place Trestle dialog box* (on page 105).

Place Equipment Dialog Box (on page 95)

Place Module Dialog Box (on page 100)

Place Trestle Dialog Box (on page 105)

What do you want to do?

- *Place idlers* (on page 90)
- *Place a pulley* (on page 91)
- *Place modules* (on page 92)
- *Place a trestle* (on page 93)
- *Modify belt object properties* (on page 95)
- *Move a belt object dynamically* (on page 95)
- *Delete a belt component* (on page 95)

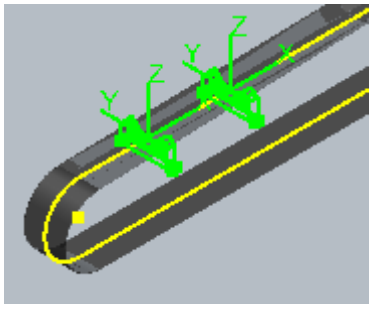
Place idlers

1. In the Material Handling task, click **Place Belt Components in 3D** on the vertical toolbar.
2. Select a belt profile, either from the **Workspace Explorer** or from the graphic view.
3. In the **Select Parent** box, select a parent system for the idlers.
4. In the **Type of Components** box, select **Equipment**.

   *The **Place Equipment** dialog box displays.*
5. In the dialog box, select the idler symbol, and select or create a system for the idlers.
6. Type the number of idlers, the offset of the first idler from the starting location, and the spacing between idlers.
7. Select the equipment orientation.

   TIP Idlers are typically placed along the carry region of a belt. **Normal to Belt** and **Vertical Up** are common selections.
8. Select a selection mode and a placement direction.
9. In the graphic view, click a starting location. Click a point for the **Point on Belt Path** selection mode, or click existing equipment for the **Existing Equipment** selection mode.
10. Click **Preview** in the **Place Equipment** dialog box.

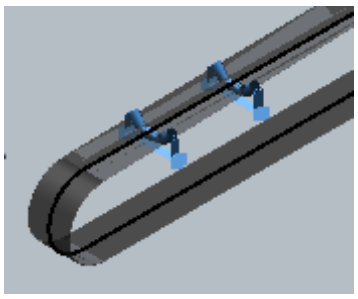
    A preview of the idlers displays.



11. To change the idlers, click **Undo**, and then make the required changes to the idler properties.
12. To place the idlers and leave the dialog box open for addition placement, click **Apply**.

13. To place the idlers and close the dialog box, click **OK**.

    The idler symbols are added to the model.



NOTE To add packing between idlers and trusses, see *Packing Thickness in 3D* (on page 113).

Place a pulley

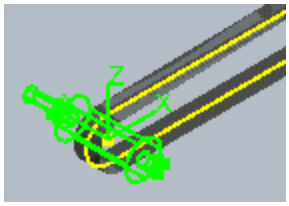
1. In the Material Handling task, click **Place Belt Components in 3D** on the vertical toolbar.
2. Select a belt profile, either from the **Workspace Explorer** or from the graphic view.
3. In the **Select Parent** box, select a parent system for the pulley.
4. In the **Type of Components** box, select **Equipment**.

   *The **Place Equipment** dialog box displays.*
5. In the dialog box, select the pulley symbol.

   *The software changes the value in the **Count** box to **1.***
6. Select or create a system.
7. Select the equipment orientation.

   TIP **Normal to Belt** is a common selection.
8. In the graphic view, click a starting location at the center or on the arc of a pulley region.
9. Click **Preview** in the **Place Equipment** dialog box.

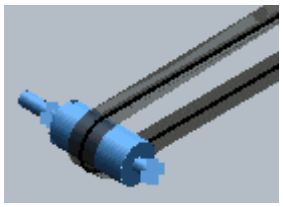
   A preview of the pulley displays.



10. To change the pulley, click **Undo**, and then make the required changes to the pulley properties.
11. To place the pulley and leave the dialog box open for addition placement, click **Apply**.

12. To place the pulley and close the dialog box, click **OK**.

    The idler symbols are added to the model.



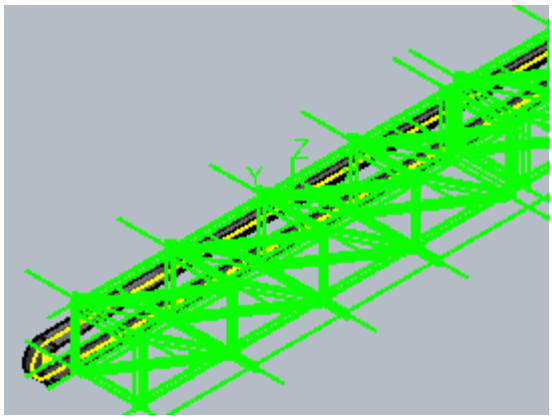
Place modules

1. In the Material Handling task, click **Place Belt Components in 3D** on the vertical toolbar.
2. Select a belt profile, either from the **Workspace Explorer** or from the graphic view.
3. In the **Select Parent** box, select a parent system for the trusses.
4. In the **Type of Components** box, select **Modules**.

   *The **Place Modules** dialog box displays.*
5. In the dialog box, select the truss module, and select or create a system for the trusses.
6. Type the number of trusses, the offset of the first truss from the starting location, and the spacing between trusses.
7. Select the module orientation.

   TIP **Normal to Belt** is a common selection.
8. Select a selection mode and a placement direction.
9. In the graphic view, click a starting location. Click a point for the **Point on Belt Path** selection mode, or click existing equipment or modules for the **Existing Module** selection mode.
10. Click **Preview** in the **Place Modules** dialog box.

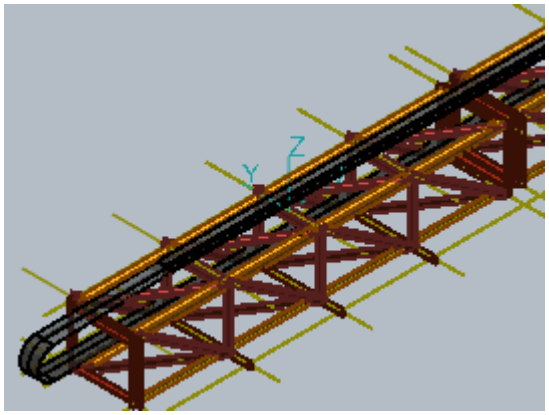
    A preview of the trusses displays.



11. To change the trusses, click **Undo**, and then make the required changes to the truss properties.

12. To place the trusses and leave the dialog box open for addition placement, click **Apply**.
13. To place the trusses and close the dialog box, click **OK**.

    The truss symbols are added to the model.



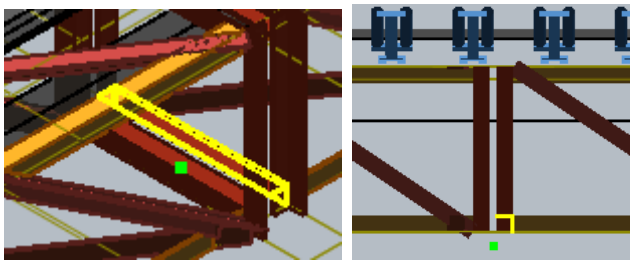
NOTE To add packing between idlers and trusses, see *Packing Thickness in 3D* (on page 113).

Place a trestle

1. (Optional) In the Grids task, add an elevation grid plane to the existing coordinate system at the required location of the bottom of the trestle.
2. In the Material Handling task, click **Place Belt Components in 3D** on the vertical toolbar.
3. Select a belt profile, either from the **Workspace Explorer** or from the graphic view.
4. In the **Select Parent** box, select a parent system for the trestle.
5. In the **Type of Components** box, select **Trestles**.

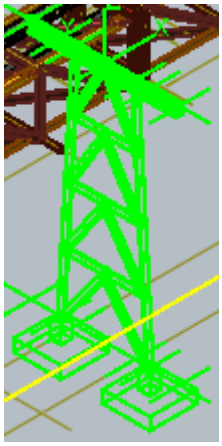
   *The **Place Trestle** dialog box displays.*
6. In the dialog box, select the trestle type (**Trestles** or **Short/Long Posts**), and select or create a system for the trestle.
7. Adjust values as required for placement parameters, and column, beam, and bracing properties.
8. In the graphic view, select a top location for the trestle in the area between two trusses.

   TIP You do not need to be precise in your selection. The software defines the correct point for selections near the end of a truss.



9. In the graphic view, select a bottom location for the trestle, such as an elevation grid plane.
10. Click **Preview** in the **Place Trestle** dialog box.

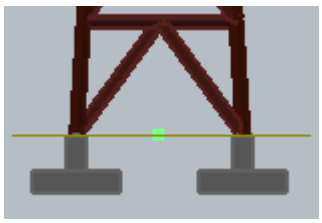
    A preview of the trestle displays.



11. To change the trestle, click **Undo**, and then make the required changes to the trestle properties.
12. To place the trestle and leave the dialog box open for addition placement, click **Apply**.
13. To place the trestle and close the dialog box, click **OK**.

    The trestle members and footing objects are added to the model.

The software places footings below the selected bottom elevation.



Modify belt object properties

1. In the Material Handling task, click **Modify Belt Objects** on the vertical toolbar.
2. Select the equipment object.
3. Click **Edit Relation Properties**.

   *The **Modify Belt Objects** dialog box displays.*
   Modify Belt Objects Dialog Box *(on page 119)*
4. Modify the properties of the object, and click **OK**.

Move a belt object dynamically

1. In the Material Handling task, click **Modify Belt Objects** on the vertical toolbar.
2. Select the equipment object.
3. Click **Dynamically Move Object**.
4. Drag the object to its new position.
5. Click to finalize the edits.

Delete a belt component

1. Click **Select** on the vertical toolbar.
2. Select **Equipment** in the **Locate Filter**.
3. Select the belt component to delete.
4. Click **Delete**.

Place Equipment Dialog Box

Specifies the type and properties for the belt equipment.

Equipment

Select Equipment

Specifies the type of equipment. Select a recently used type, or click **More** to display the **Select Equipment** dialog box. In the **Material Handling Equipment** folder, select a folder such as **Conveyor Drives**, **Conveyor Pulleys**, or **Idlers**, and then select an equipment part from the right pane of the dialog box.

NOTE Click **Preview** on the **Select Equipment** dialog box ribbon to display a preview of the selected equipment part.

Select Subsystem

Specifies a sub-system within the parent system. Select an existing system, or click **Create**

New Subsystem to display the **New Parent Subsystem** dialog box and create a new sub-system.

NOTE The new system is a child to the parent system defined in **Select Parent** on the ribbon.

Placement Parameters

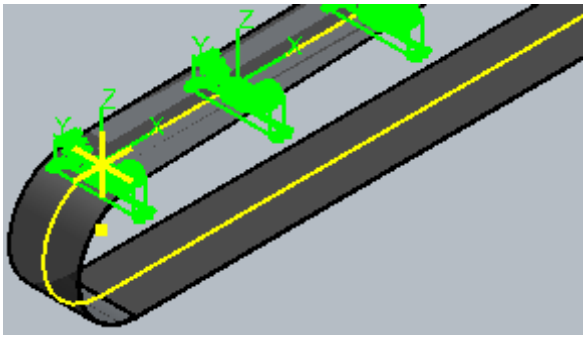
Count

Defines the quantity of items to place.

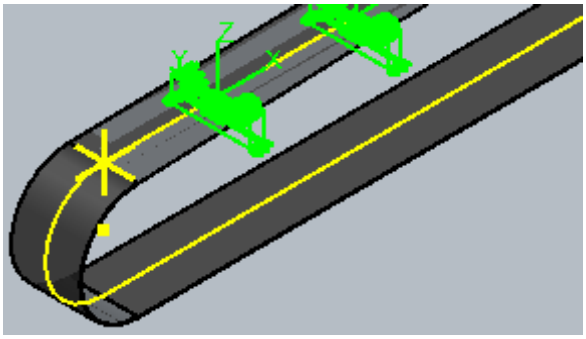
Offset

Defines the distance of placement of the center of the first equipment symbol from the starting point selected based on the method defined in **Selection Mode**. See the following examples.

Offset of 0 mm from the starting point:



Offset of 1000 mm from the starting point:



Spacing

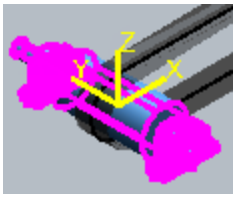
Defines the center-to-center distance between equipment items along the belt profile.

Equipment Orientation

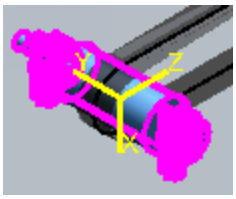
Specifies the orientation of the equipment. All orientations are based on the position of the local z-axis of the equipment symbol. The positive z-axis represents the up position of an equipment symbol.

NOTE An unsymmetrical conveyor drive is shown in the following examples.

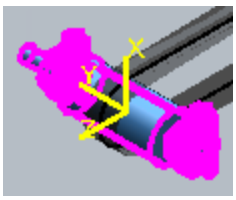
- **Normal to Belt** - Orients the local z-axis of the symbol normal to the belt profile. The symbol remains normal to the belt regardless of the belt orientation, and the local x-axis is tangent to the belt profile path. The orientation angle is **0 deg**.



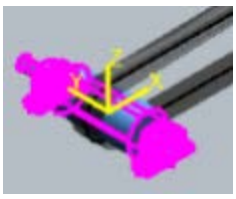
- **Horizontal Right** - Rotates the local z-axis of the symbol 90° about the y-axis. The symbol remains horizontal with respect to the global coordinate system regardless of the belt orientation. The orientation angle is **90 deg**.



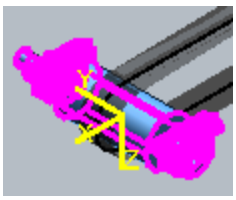
- **Horizontal Left** - Rotates the local z-axis of the symbol 270° about the y-axis. The symbol remains horizontal with respect to the global coordinate system regardless of the belt orientation. The orientation angle is **270 deg**.



- **Vertical Up** - No rotation of the local z-axis. The symbol remains vertical with respect to the global coordinate system regardless of the belt orientation. The orientation angle is **0 deg**.



- **Vertical Down** - Rotates the local z-axis of the symbol 180° about the y-axis. The symbol remains vertical with respect to the global coordinate system regardless of the belt orientation. The orientation angle is **180 deg**.

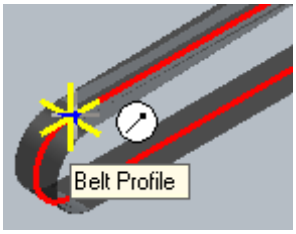


- **Key to Angle** - Rotates the local z-axis of the symbol the specified angle about the y-axis. Type the required orientation angle.

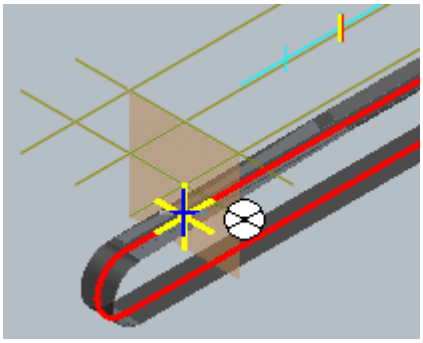
Selection Mode

Specifies the method of defining the starting location of equipment or module placement.

- **Point on Belt Path** - Select a point on the conveyor in the graphic view, such as:
  - A point at the end of a belt region, or at the end or the center of a pulley region.

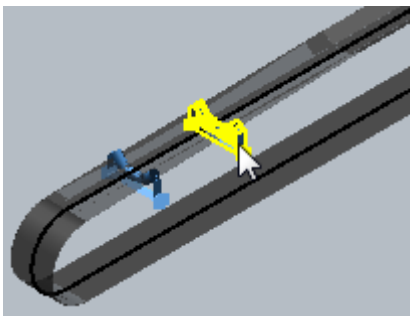


  - A point at the intersection of the plane and the belt path.



NOTE Select the plane, and then select a point at the intersection of the plane and the belt.

- **Existing Equipment** - Select an existing equipment placed on the conveyor. Select the equipment in the graphic view.

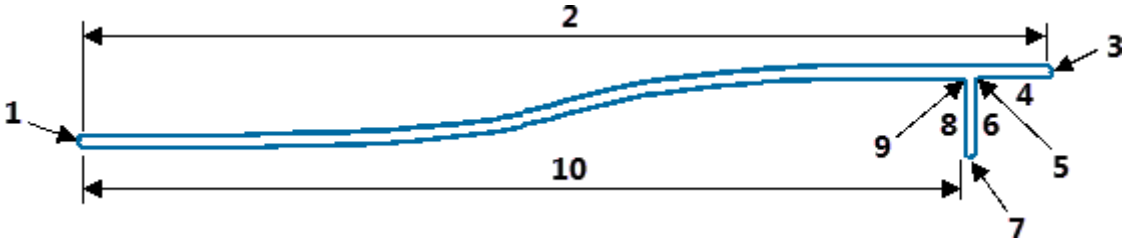


Placement Direction

Specifies the direction of equipment or module placement.

- **Tail End to Head End** - Placement starts at the end closest to the tail pulley and progresses towards the head pulley.

- **Head End to Tail End** - Placement starts at the end closest to the head pulley and progresses towards the tail pulley.

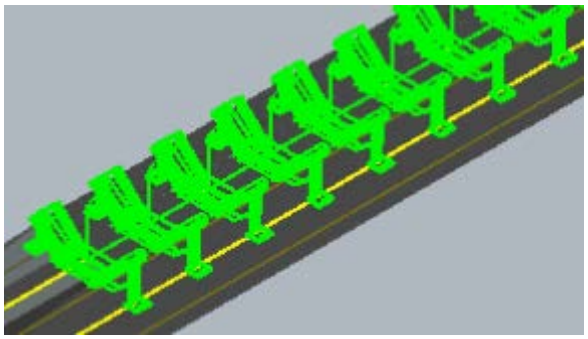


1 - Tail pulley region
2 - Carry region
3 - Head pulley region
5, **7**, **9** - Pulley regions
4, **6**, **8**, **10** - Belt regions

NOTE Idlers, drives, trusses, and trestles are typically placed on the carry region.

Preview

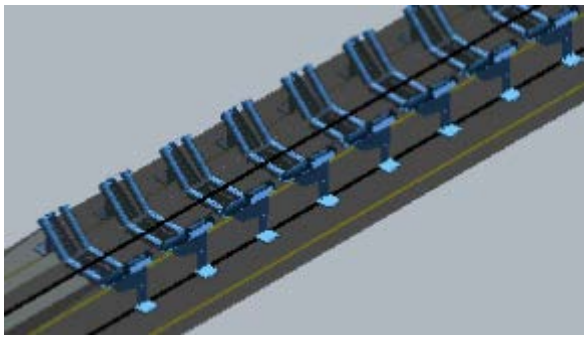
Displays a preview of the equipment.



NOTE You can click **Undo** to clear the preview, change any values on the dialog box, and then click **Preview** again to see an updated preview.

Apply

Places the equipment on the belt. The dialog box remains open, and you can place additional equipment.



NOTE The **Existing Equipment** selection mode is commonly used to select the starting location for additional equipment.

Undo

Removes the last **Preview** results.

OK

Places the items on the belt, and closes the dialog box.

Cancel

Cancels the last set of property definitions, and closes the dialog box.

NOTE Items shown in an existing preview are not added to the model. Items added by clicking **Apply** remains in the model.

Place Module Dialog Box

Specifies the type and properties for the truss supporting the belt and equipment or other belt module.

Module

Select Module

Specifies the type of module. Select a recently used type, or click **More** to display the **Select Module** dialog box. In the **Modules** folder, select a folder, and then select a module from the right pane of the dialog box.

NOTES

- Click **Preview** on the **Select Module** dialog box ribbon to display a preview of the selected module part.
- Truss modules are available in the **Conveyor System** folder.
- If no modules display in the right pane, your system administrator must save them to the catalog. For example, an administrator can use the **Copy to Catalog** option of **Truss Wizard** to save a truss to the catalog. For more information, see *General Tab (Truss Wizard)* (on page 64).

Select Subsystem

Specifies a sub-system within the parent system. Select an existing system, or click **Create New Subsystem** to display the **New Parent Subsystem** dialog box and create a new sub-system.

NOTE The new system is a child to the parent system defined in **Select Parent** on the ribbon.

Placement Parameters

Count

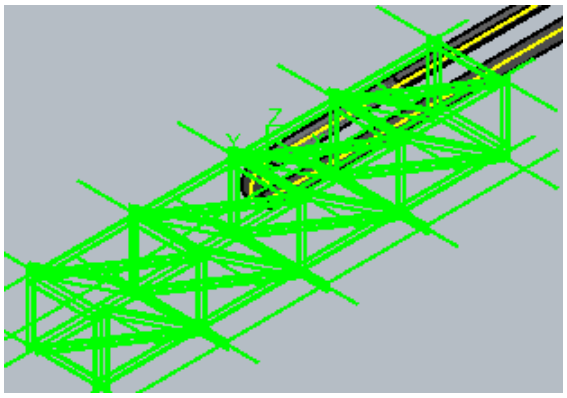
Defines the quantity of items to place.

Offset

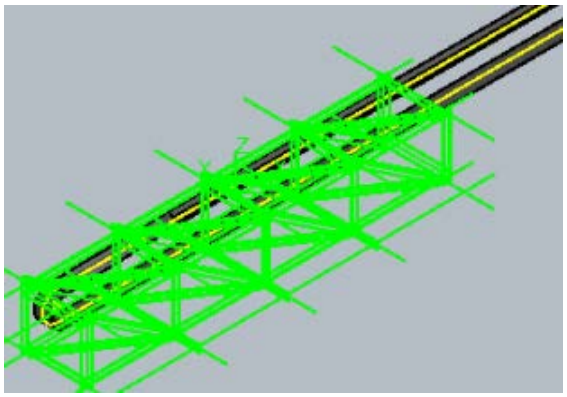
Defines the distance of placement of the center of the first module symbol from the starting point selected based on the method defined in **Selection Mode**.

TIP For trusses, consider one-half the length of the truss in the offset. For example, if the trusses are 12m long, then you might want a minimum offset of 6m. See the following examples.

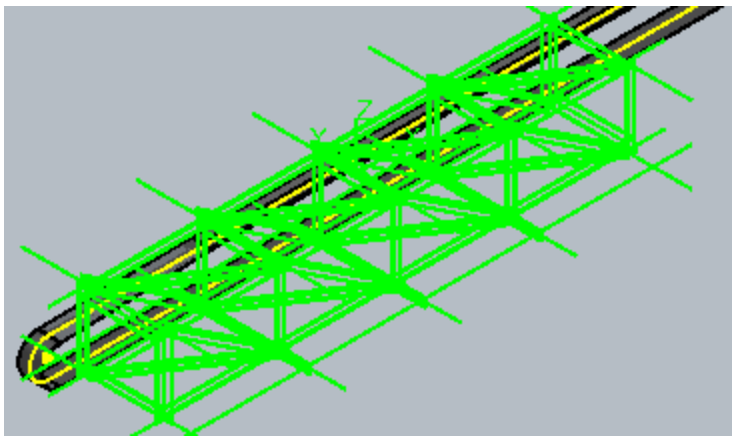
12m truss offset of 0 m from the starting point:



12m truss offset of 6m from the starting point:



12m truss offset of 8m from the starting point:



Spacing

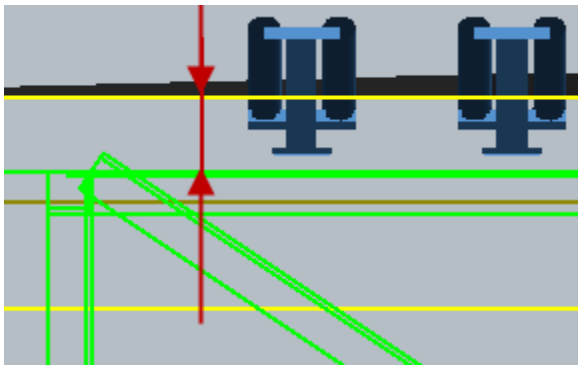
Defines the center-to-center distance between module items along the belt profile.

TIP For trusses, include the gap required between each truss in the spacing. For

example, if the trusses are 12m long and need a 0.5m gap, then the spacing is 12.5m.

Idler Height

Specifies the height of the idler plus the thickness of packing between the idler and the truss.

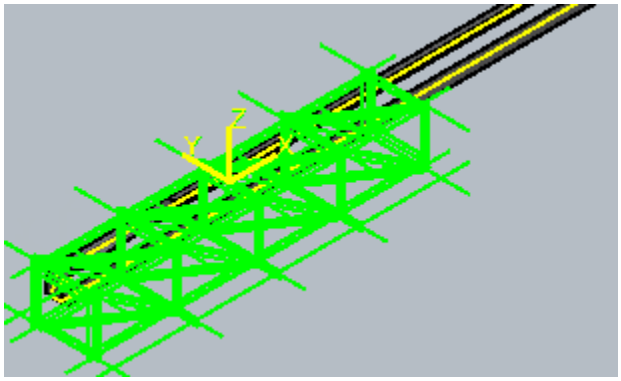


NOTE The height of the idler is the distance from the top of the bottom roller to the bottom of the idler base plate.

Module Orientation

Specifies the orientation of the module. All orientations are based on the position of the local z-axis of the module symbol. The positive z-axis represents the up position of a module symbol.

Normal to Belt - Orients the local z-axis of the symbol normal to the belt profile. The symbol remains normal to the belt regardless of the belt orientation, and the local x-axis is tangent to the belt profile path. The orientation angle is **0 deg**.



Horizontal Right - Rotates the local z-axis of the symbol 90° about the y-axis. The symbol remains horizontal with respect to the global coordinate system regardless of the belt orientation. The orientation angle is **90 deg**.

NOTE Do not use this option for a truss.

Horizontal Left - Rotates the local z-axis of the symbol 270° about the y-axis. The symbol remains horizontal with respect to the global coordinate system regardless of the belt orientation. The orientation angle is **270 deg**.

NOTE Do not use this option for a truss.

Vertical Up - No rotation of the local z-axis. The symbol remains vertical with respect to the global coordinate system regardless of the belt orientation. The orientation angle is **0 deg**.

NOTE Do not use this option for a truss.

Vertical Down - Rotates the local z-axis of the symbol 180° about the y-axis. The symbol remains vertical with respect to the global coordinate system regardless of the belt orientation. The orientation angle is **180 deg**.

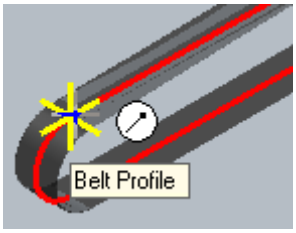
NOTE Do not use this option for a truss.

Key to Angle - Rotates the local z-axis of the symbol the specified angle about the y-axis. Type the required orientation angle.

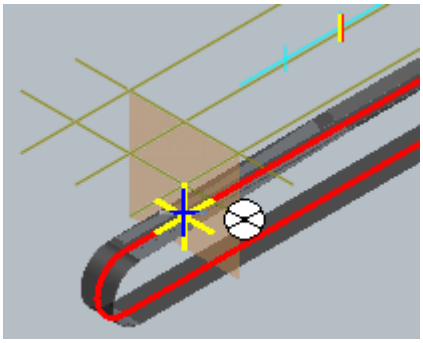
Selection Mode

Specifies the method of defining the starting location of equipment or module placement.

- **Point on Belt Path** - Select a point on the conveyor in the graphic view, such as:
  - A point at the end of a belt region, or at the end or the center of a pulley region.

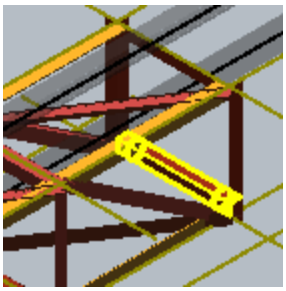


  - A point at the intersection of the plane and the belt path.



NOTE Select the plane, and then select a point at the intersection of the plane and the belt.

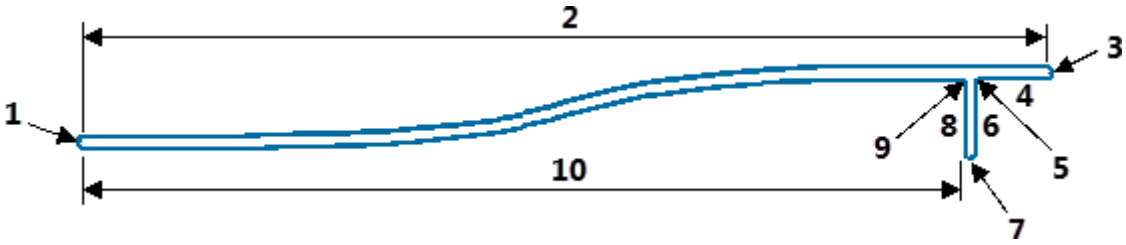
- **Existing Module** - Select a member system on an existing truss.



Placement Direction

Specifies the direction of equipment or module placement.

- **Tail End to Head End** - Placement starts at the end closest to the tail pulley and progresses towards the head pulley.
- **Head End to Tail End** - Placement starts at the end closest to the head pulley and progresses towards the tail pulley.

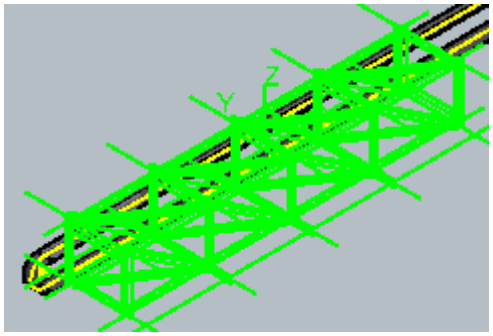


1 - Tail pulley region
2 - Carry region
3 - Head pulley region
5, **7**, **9** - Pulley regions
4, **6**, **8**, **10** - Belt regions

NOTE Idlers, drives, trusses, and trestles are typically placed on the carry region.

Preview

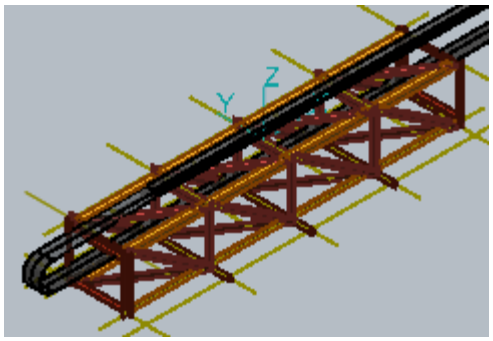
Displays a preview of the module.



NOTE You can click **Undo** to clear the preview, change any values on the dialog box, and then click **Preview** again to see an updated preview.

Apply

Places the module on the belt. The dialog box remains open, and you can place additional modules.



NOTE The **Existing Module** selection mode is commonly used to select the starting

location for additional modules.

Undo

Removes the last **Preview** results.

OK

Places the items on the belt, and closes the dialog box.

Cancel

Cancels the last set of property definitions, and closes the dialog box.

NOTE Items shown in an existing preview are not added to the model. Items added by clicking **Apply** remains in the model.

Place Trestle Dialog Box

Specifies the type and properties for a trestle supporting the belt truss.

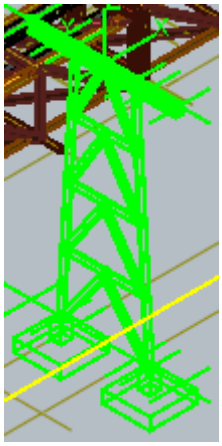
Place Tab (Place Trestle Dialog Box) (on page 106)

Columns and Top Beam Tab (Place Trestle Dialog Box) (on page 109)

Bracing and Cross Beams Tab (Place Trestle Dialog Box) (on page 110)

Preview

Displays a preview of the support.



NOTE You can click **Undo** to clear the preview, change any values on the dialog box, and then click **Preview** again to see an updated preview.

Apply

Places the support on the belt. The dialog box remains open, and you can place additional supports.

NOTE The **Existing Symbol** selection mode is commonly used to select the starting location for additional supports.

Undo

Removes the last **Preview** results.

OK

Places the items on the belt, and closes the dialog box.

Cancel

Cancels the last set of property definitions, and closes the dialog box.

NOTE Items shown in an existing preview are not added to the model. Items added by clicking **Apply** remains in the model.

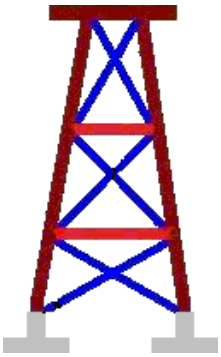
Place Tab (Place Trestle Dialog Box)

Specifies the type and dimensional properties for the trestle.

Select Trestle Type

Specifies the type of support.

- **Trestles** - Support consisting of columns, cross bracing, and a top beam.



- **Short/Long Posts** - Support consisting of columns and a top beam.

Select Subsystem

Specifies a sub-system within the parent system. Select an existing system, or click **Create New Subsystem** to display the **New Parent Subsystem** dialog box and create a new sub-system.

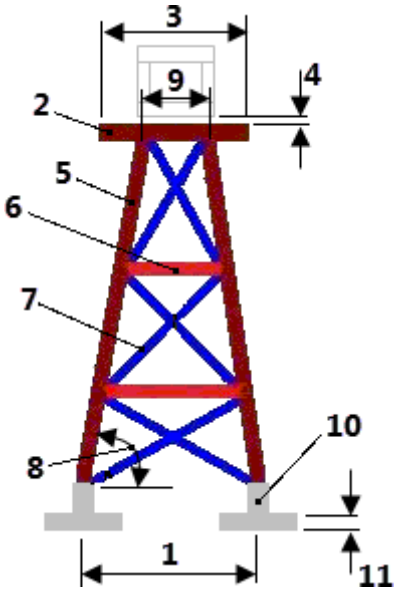
NOTE The new system is a child to the parent system defined in **Select Parent** on the ribbon.

Preview

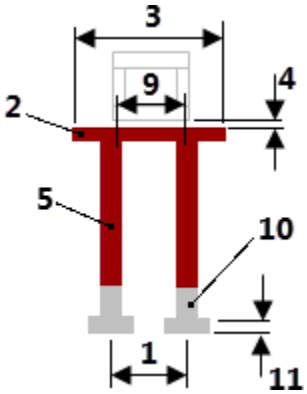
Displays a preview of the selected support type.

Placement Parameters

Trestles:



Posts:



1 - Base Width
2 - Top Beam
3 - Top Beam Length
4 - Clearance Height
5 - Column or Post
6 - Cross Beam
7 - Bracing
8 - Angle
9 - Working Point Length
10 - Footing
11 - Footing Depth

Base Width

Specifies the center-to-center distance between the columns at the base of the trestle.

Top Beam Length

Specifies the length of the top beam supporting the trusses.

Clearance Height

Specifies the distance between the bottom of the truss and the top of the top beam at the

selection point of trestle placement.

No. of Cross Beams

Specifies the quantity of horizontal beams providing lateral support to the trestle. This option is not available when you select **Short/Long Posts** as the trestle type.

Angle

Specifies the angle of the trestle columns. Select **Angle** or **Working Point**. This option is not available when you select **Short/Long Posts** as the trestle type.

Working Point

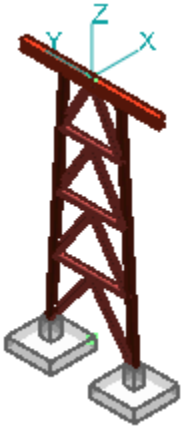
Specifies the center-to-center between the columns at the top of the trestle. Select **Angle** or **Working Point**. This option is not available when you select **Short/Long Posts** as the trestle type.

Display Grid Lines

When selected, displays the grid lines used to define the trestle.

Display Grid Lines selected:

Display Grid Lines cleared:



Footing

Specifies whether a footing is created for the trestle.

When a footing is specified, you then select the type of footing. Select a recently used type, or click **More** to display the **Select Equipment** dialog box. In the **Assemblies** folder, select the **Combined** or **Single** folder, and then select a footing from the right pane of the dialog box.

NOTE Click **Preview** on the **Select Equipment** dialog box ribbon to display a preview of the selected footing symbol.

Depth

Specifies the depth of the footing. Depth cannot be customized for all footings.

Columns and Top Beam Tab (Place Trestle Dialog Box)

Specifies the shape and material properties for the column and top beam members of the trestle.

Columns

Member Type

Specifies the type of member. The default value is **Column**.

Standard

Displays the structural specification defined in the catalog.

Section

Specifies the structural cross-section shape. Select a recently used type, or click **More** to display the **Catalog Browser** dialog box. In the **Shapes** folder, select a folder, and then select a module from the right pane of the dialog box.

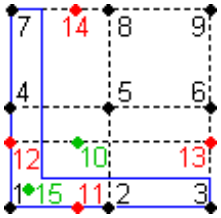
Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

Specifies the object material grade, such as **A36** or **A529**.

Cardinal Point



Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

Angle

Specifies the rotation of the cross-section about its local axis.

Top Beam

Member Type

Specifies the type of member. The default value is **Beam**.

Standard

Displays the structural specification defined in the catalog.

Section

Specifies the structural cross-section shape. Select a recently used type, or click **More** to display the **Catalog Browser** dialog box. In the **Shapes** folder, select a folder, and then select a module from the right pane of the dialog box.

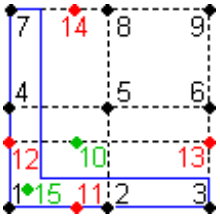
Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

Specifies the object material grade, such as **A36** or **A529**.

Cardinal Point



Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

Angle

Specifies the rotation of the cross-section about its local axis.

Bracing and Cross Beams Tab (Place Trestle Dialog Box)

Specifies the shape and material properties for the bracing and cross beam members of the trestle.

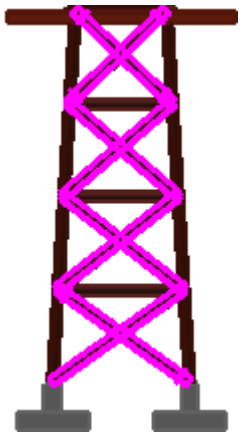
NOTE This tab is not available when you select **Short/Long Posts** as the trestle type.

Bracing

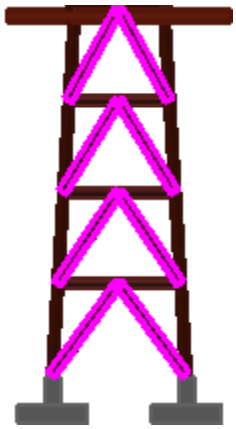
Bracing Type

Specifies the bracing configuration. Select one of the following options:

- **Cross**



- **Chevron**



Member Type

Specifies the type of member. The default value is **Brace**.

Standard

Displays the structural specification defined in the catalog.

Section

Specifies the structural cross-section shape. Select a recently used type, or click **More** to display the **Catalog Browser** dialog box. In the **Shapes** folder, select a folder, and then select a module from the right pane of the dialog box.

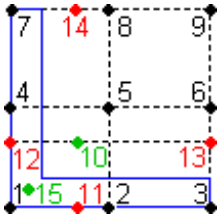
Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

Specifies the object material grade, such as **A36** or **A529**.

Cardinal Point



Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

Angle

Specifies the rotation of the cross-section about its local axis.

Cross Beams

Member Type

Specifies the type of member. The default value is **Brace**.

Standard

Displays the structural specification defined in the catalog.

Section

Specifies the structural cross-section shape. Select a recently used type, or click **More** to display the **Catalog Browser** dialog box. In the **Shapes** folder, select a folder, and then select a module from the right pane of the dialog box.

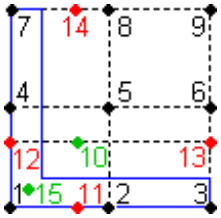
Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

Specifies the object material grade, such as **A36** or **A529**.

Cardinal Point



Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

Angle

Specifies the rotation of the cross-section about its local axis.

SECTION 11

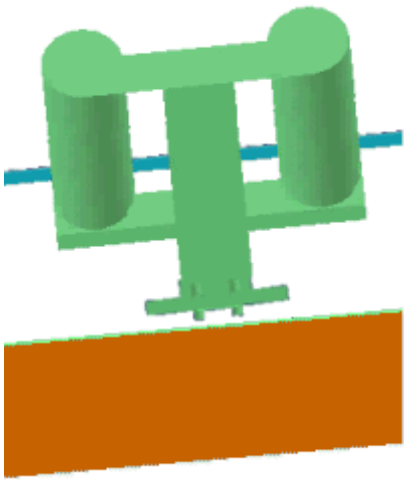
Packing Thickness in 3D

Places additional packing material thickness below idlers so that they mount perfectly on the trusses. The software automatically calculates the required packing thickness for the 3D idler symbols in the model. The packing thickness can be different for each idler.

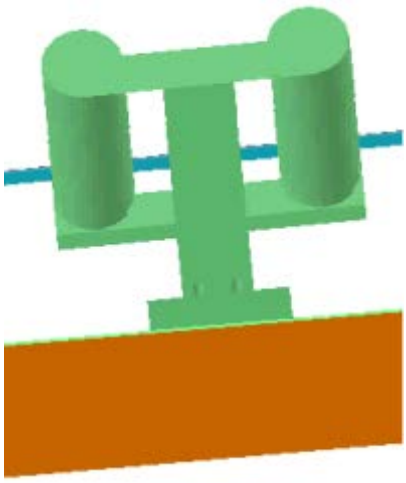
NOTE A packing plate is a distinct object type in the model and has no relationship to plates created in the Molded Forms task.

Additional packing is generally required when the belt transitions between a linear and curved profile and the distances vary between idlers and trusses.

Before packing:



After adding packing:



Reports

Use the **Tools > Run Reports** command to generate an idler packing thickness report. For more information about reports, see *Run Report* in the *Common User's Guide*.

Packing Thickness in 3D Ribbon

Select Truss

Specifies the truss to associate with idlers.

Add/Remove

Adds or removes the selected idlers. No packing is added to a removed idler.

NOTE You should only add or remove idlers associated with the selected truss. Do not add idlers from other trusses.

Accept

Sets packing thickness attributes on the idlers.

Cancel

Removes the packing thickness attributes from the idlers.

Finish

Writes your changes to the model.

What do you want to do?

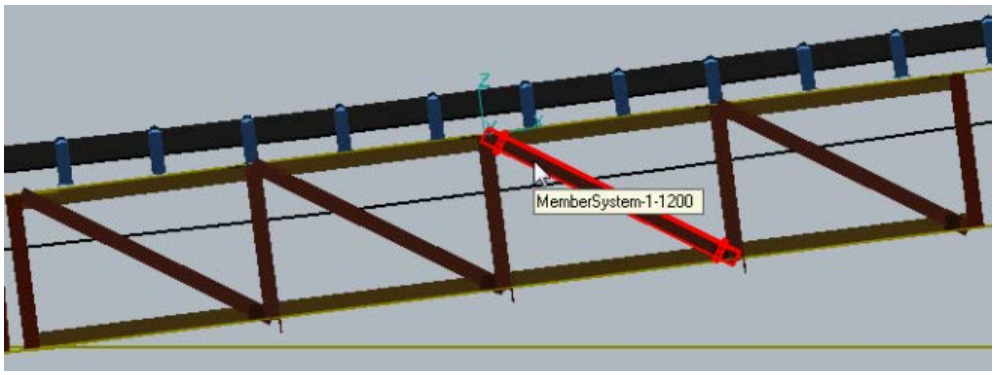
- *Add packing thickness in 3D* (on page 114)
- *Remove packing thickness in 3D* (on page 116)
- *Create an idler packing thickness report* (on page 183)

Add packing thickness in 3D

1. Click **Packing Thickness in 3D** on the vertical toolbar.

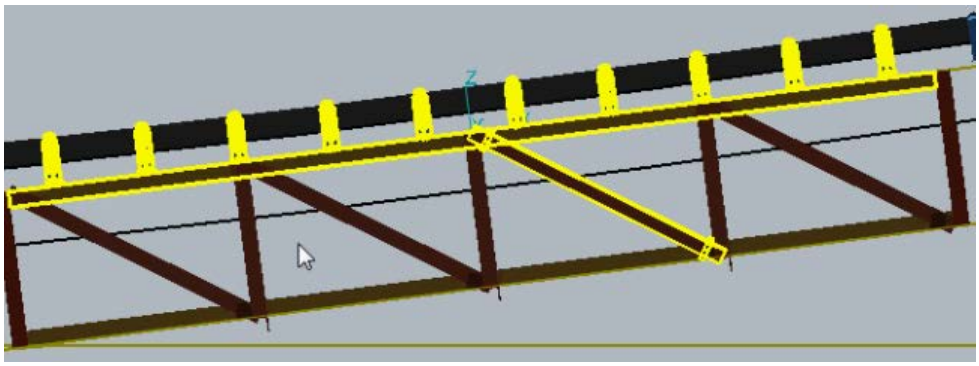
   *The **Packing Thickness in 3D** ribbon displays, and Smart 3D.prompts you to select a member or parent system for the module.*

2. Select a member or parent system associated with the idlers.



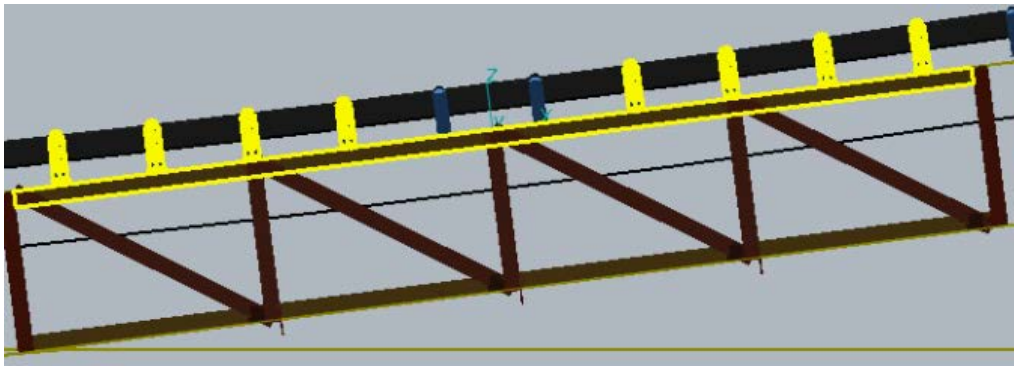
TIP You can select from either the model or the **Workspace Explorer**.

Smart 3D highlights the truss and all of the idlers.



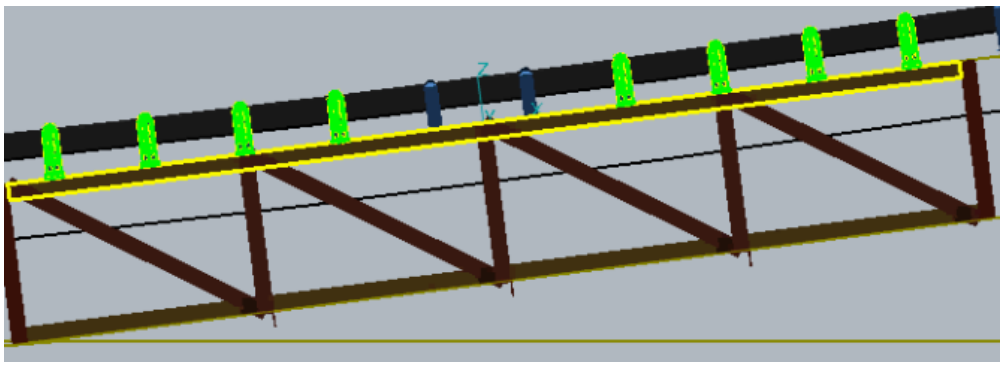
3. Click **Add/Remove**.

4. Select idlers to remove as required.

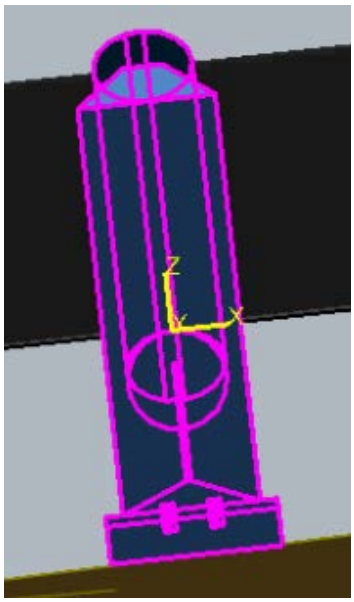


Smart 3D highlights or clears the highlight from the idlers as you select them. The software adds packing thickness only to the highlighted idlers.

5. Click **Accept** ✓ to set the packing thickness on the selected idlers.



6. Click **Finish**.



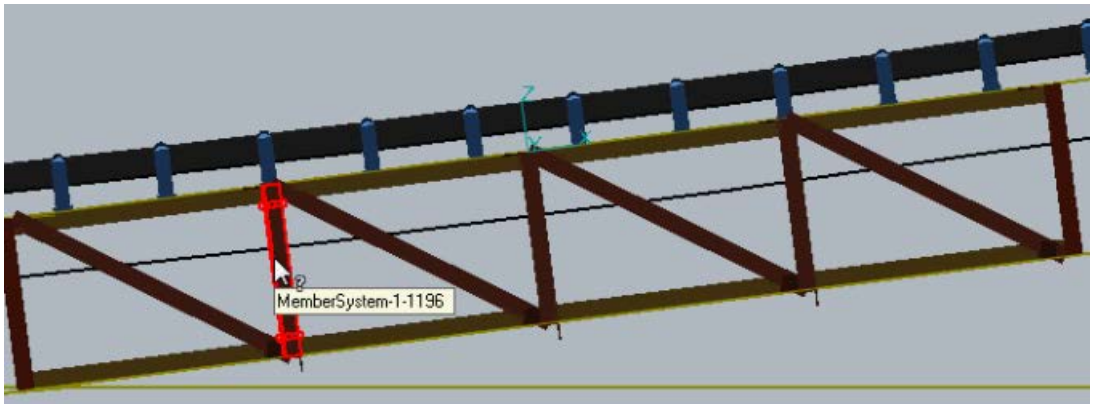
Smart 3D adds packing between each idler and the truss.

Remove packing thickness in 3D

1. Click **Packing Thickness in 3D** on the vertical toolbar.

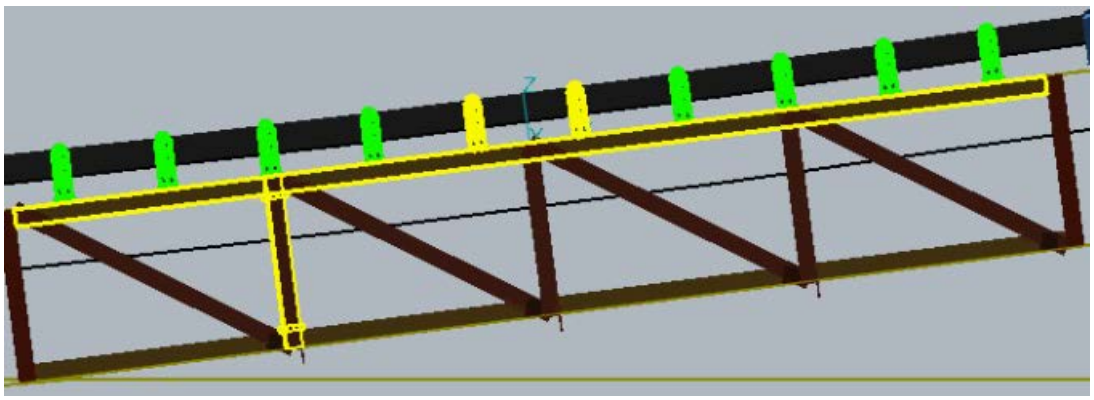
   *The **Packing Thickness in 3D** ribbon displays, and Smart 3D.prompts you to select a member or parent system for the module.*

2. Select a member or parent system associated with the idlers.



TIP You can select from either the model or the **Workspace Explorer**.

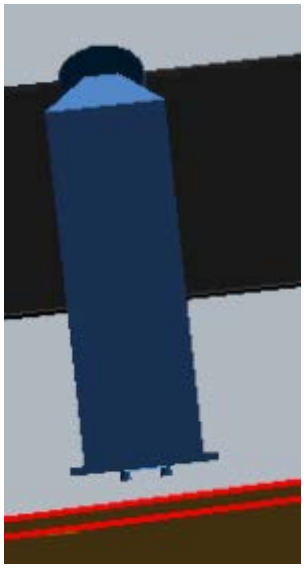
Smart 3D highlights the truss and all of the idlers. Idlers with packing applied display in green. Idlers with no packing applied display in yellow.



3. Click **Cancel** to remove the packing thickness.

4. Click **Finish**.

   Smart 3D removes the packing thickness from all of the idlers.



SECTION 12

Modify Belt Objects

Modifies all controlled objects that have a parametric relationship to a belt. Most modify commands, such as **Move** and **Rotate**, cannot be used to alter the position or orientation of objects that have a parametric relation to a belt.

Modify Belt Objects Ribbon

Select Object

Selects a belt object.

Edit Relation Properties

Modifies the relationship properties of a belt object.

Dynamically Move Object

Dynamically moves the belt object.

What do you want to do?

- *Modify a belt object's relationship to a belt* (on page 118)
- *Modify belt object properties* (on page 95)
- *Move a belt object dynamically* (on page 95)

Modify a belt object's relationship to a belt

1. In the Material Handling task, click **Modify Belt Objects** on the vertical toolbar.
2. Select the object to modify.
3. To modify an equipment object on a conveyor belt, select the equipment object directly. To modify a module, select any member or the parent system of the module from the graphic view or the **Workspace Explorer**.

Modify belt object properties

1. In the Material Handling task, click **Modify Belt Objects** on the vertical toolbar.
2. Select the equipment object.
3. Click **Edit Relation Properties**.

   *The **Modify Belt Objects** dialog box displays.*
   Modify Belt Objects Dialog Box *(on page 119)*
4. Modify the properties of the object, and click **OK**.

Move a belt object dynamically

1. In the Material Handling task, click **Modify Belt Objects** on the vertical toolbar.
2. Select the equipment object.
3. Click **Dynamically Move Object**.
4. Drag the object to its new position.
5. Click to finalize the edits.

Modify Belt Objects Dialog Box

Name

Displays the name of the object.

Relation Type

Displays the relation that the object has to the belt:

- **Aligned** - The object x-axis is tangent to the direction of the belt.
- **Angle** - The object x-axis is angled based on the horizontal direction of the belt. The angle value is defined by **Angle**.
- **Arc center** - The object is a pulley, or an object such as a head or tail frame that should be aligned with a pulley axis. The angle of the placed object is defined by **Angle**.

Relation Flags

Defines the position of the object to the belt:

- **Outer surface** - The object is attached to the outside surface of the belt.
- **Z upwards** - The object z-axis is upwards, preventing the object from being placed upside-down on the belt.
- **Offset upwards** - The object offset is upwards.
- **Reverse direction** - The object is a return idler.

Distance

Defines the distance between two reference points on the belt. Each reference point represents a pulley on the belt. A negative value represents the distance from the end reference point.

Offset

Defines the distance from the object to the belt. For the carry sections of the belt, the object is offset in an upward direction from the belt. For the return sections of the belt, the object is offset in a downward direction from the belt.

Angle

Defines the angle of rotation of the object to the belt. This angle is counter-clockwise.

Apply

Applies the values to the object.

OK

Applies the values to the object.

Reset

Discards any changes that have not yet been finalized by clicking **OK**.

Close

Closes the **Modify Belt Objects** dialog box.

SECTION 13

Create Control Points

Places one or more control points anywhere in the workspace, including at grid intersections.

The Material Handling task uses control points in general arrangement drawings of chutes, conveyors, gantries, and trusses.

When you create a control point, you must add it to the **System** hierarchy in the **Workspace Explorer** beneath a generic system. For more information see *New Generic System* in the *Systems and Specifications User's Guide*.

Create Control Points is similar to **Insert** > **Control Point**, except that it allows:

- Placement of more than one control point at a time
- More flexibility in where control points are placed

Create Control Point Ribbon

Properties

Opens the **Control Point Properties** dialog box. You can use this dialog box to define the standard properties of the control point during initial placement or modify standard properties after placement. For more information, see *Control Point Properties Dialog Box* (on page 124).

Select Parent

Select a parent generic system for the control points. Select the system in the Workspace Explorer.

Select Point Location

Allows you to click in the model to specify the location of the control point.

Type

Specifies the appropriate category of point. Select **Control Point**, **Insertion Point**, and **Keypoint**.

Subtype

Select the subtype depending upon where the control point is to be used. In the Material Handling task, most control points are restricted to **Structure**.

Name Prefix

Type a prefix name for the control point. The name will be prepended to the numeric value of the control points counter stored in the session preferences.

Constraint

Select the constraint for placement of control point. Select **None** to create control points anywhere in the workspace. Select **Grid Line Intersection** to create control points at grid intersections only.

What do you want to do?

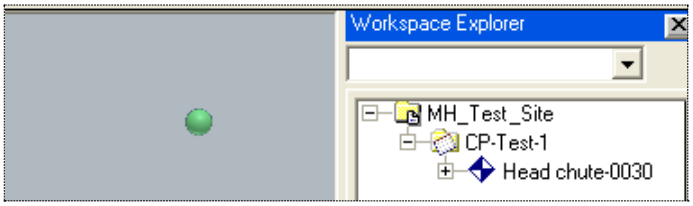
- *Create control points using the mouse* (on page 122)
- *Create control points at grid intersections* (on page 123)

Create control points using the mouse

1. If needed, create a new generic system in the Systems and Specifications task. For more information see *New Generic System* in the *Systems and Specifications User's Guide*.
2. In the Material Handling task, click **Create Control Points** on the vertical toolbar.
3. In the **Workspace Explorer**, select a generic system as the parent system.

   TIP The parent system can be any generic system.
4. Set the **Type** to **Control Point**.
5. Set the **Subtype** to **Structure**.
6. Type a value for **Name Prefix**.
7. In the **Constraint** list, select **None**.
8. Click a location in the graphic view to create the control point.

   *The control point appears in the graphic view and in the **Workspace Explorer**.*



9. Continue clicking in the graphic view to place as many control points as needed.
10. Right-click in the graphic view to exit the command.

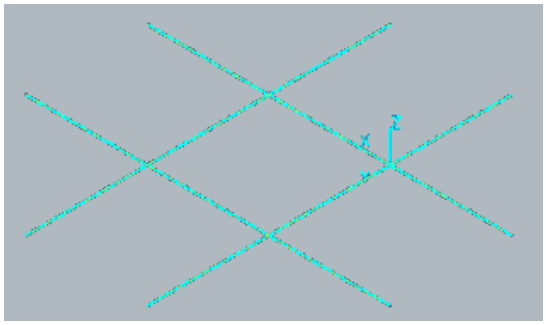
TIPS

- Use Sketch 2D commands available within Material Handling to precisely place control points. For more information, see *PinPoint, Add to Sketch 2D List*, *Get Point Point Along*, and *Measure* in the *Common User's Guide*.
- Before placing a control point, you can change the parent system at any time by selecting a new generic system in the **Workspace Explorer**.

Create control points at grid intersections

1. If needed, create a new generic system in the Systems and Specifications task. For more information, see *New Generic System* in the *Systems and Specifications User's Guide*.
2. In the Grids task, create the needed grid planes and grid lines. For more information, see the *Grids User's Guide*.

   Example:

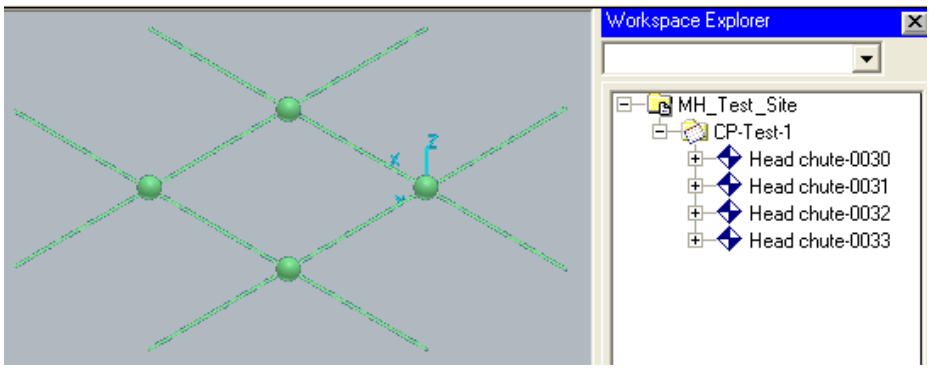
   

3. In the Material Handling task, click **Create Control Points** on the vertical toolbar.
4. In the **Workspace Explorer**, select a generic system as the parent system.

   TIP The parent system can be any generic system.
5. Set the **Type** to **Control Point**.
6. Set the **Subtype** to **Structure**.
7. Type a value for **Name Prefix**.
8. In the **Constraint** list, select **Grid Line Intersection**.
9. Click the intersection of any two grid lines in the graphic view to create the control point.

   *The control point appears in the graphic view and in the **Workspace Explorer**.*
10. Continue clicking at grid intersections to place as many control points as needed.

    Example:

11. Right-click in the graphic view to exit the command.

TIPS

- A control point is not created if you click a location that is not a grid intersection.
- Before placing a control point, you can change the parent system at any time by selecting a new generic system in the **Workspace Explorer**.

Control Point Properties Dialog Box

Sets options for a selected control point.

NOTE When you open the **Control Point Properties** dialog box during the initial placement of a control point, only the **General** tab is visible. The remaining tabs display only when you are editing an existing control point.

General Tab (Control Point Properties Dialog Box) (on page 124)

Relationship Tab (on page 32)

Configuration Tab (on page 33)

Notes Tab (on page 34)

General Tab (Control Point Properties Dialog Box)

The **General** tab displays the control point properties that you can edit or that are automatically determined by the software at placement. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one control point and then selected the **Properties** command, only the common properties between the selected control points display.

When viewing properties for a single control point, the following properties display. More properties may display depending on what you defined in the reference data. Refer to the *Material Handling Reference Data Guide* for more information on properties.

Category

Select the type of properties that you want to view for the selected control point. Control point properties have only one category: **Standard**.

Control Point Type

Specifies the appropriate category of the point. You select an option from a select list, which includes **Control Point**, **Keypoint**, and **Insertion Point**.

Control Point Subtype

Provides a further breakdown for the type of control point. In the Material Handling task, control points are usually restricted to **Structure**.

Name

Identifies the control point with a name that you can define, or use the selection that a rule defines. The **GenericNamingRules.xls** workbook lists the naming rules used in the software. For more information about naming rules, see the *Reference Data Guide.*

Naming Rule

Specifies the name rule used to generate the name of the grid plane. Set this option to **User Defined** if you want to specify the name yourself.

Diameter

Sets the diameter of the control point sphere.

Parent Object

Specifies the object to which the control point is associated. This value is read-only.

Associativity

Specifies whether the control point is associated with another object. In the Material Handling task, most control points are associated with a set of grids.

E

Displays the coordinate of the control point along the E-axis (East).

N

Displays the coordinate of the control point along the N-axis (North).

EL

Displays the coordinate of the control point along the EL-axis (Elevation).

SECTION 14

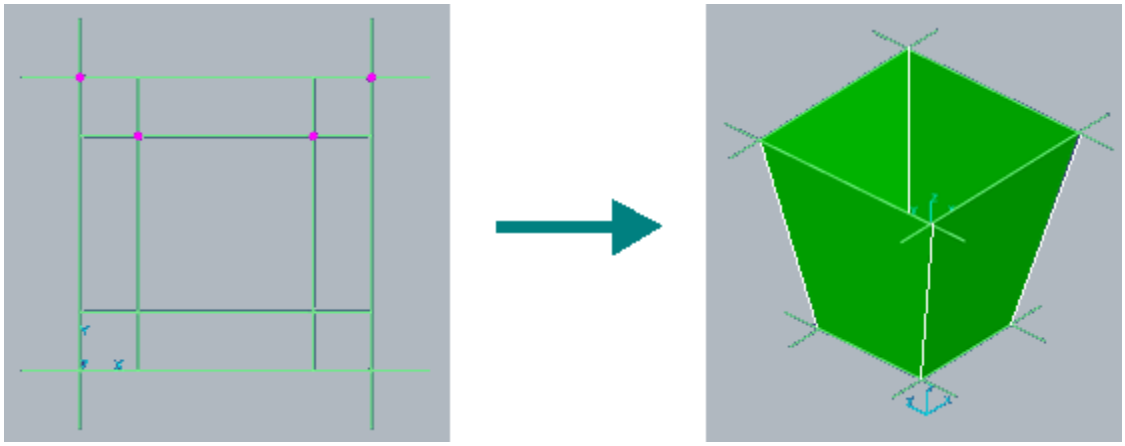
Create Chute

 Places a material handling chute using control points. You can use from three to six control points to define the end openings of the chute. The points for each opening can be the same or on different planes; however, you must have the same number of control points on each opening. In general, chutes converge down so that the lower opening is smaller than the upper opening.

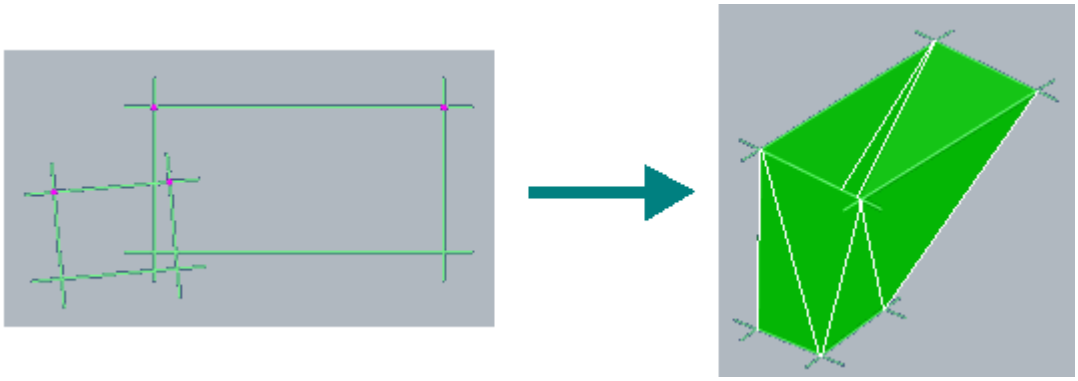
Before using this command, you must:

1. Create the grids needed to define the geometry of the chute. When placing the grids, make sure that at least three grid intersections are available for the creation of control points on each opening. For more information, see *Grids User's Guide*.
2. Place control points at the grid intersections for the top and bottom openings. Be sure to place the control points in either a clockwise or anti-clockwise direction, using the same direction for both openings. For more information, see *Create Control Points* (on page 121).

When control points create parallel upper and lower edges, a single planar plate system is created on each side of the chute. In the following example, the chute has four plates:



When control points do not create parallel upper and lower edges, two planar pate systems are created on each side of the chute. In the following example, the chute has eight plates:



★IMPORTANT **Create Chute** is only used for initial creation of the chute. After the chute is created, chute structure is placed as planar plate systems from the Molded Forms task.

Create Chute Ribbon

Specifies the properties for the chute that you are placing or editing.

Chute Plate System Properties

Activates the **Chute Properties** dialog box. You can use this dialog box to specify additional chute plate properties. For more information see *Chute Plate System Properties Dialog Box* (on page 149).

Select Top Vertices

Select the control points that define the upper face of the chute.

Select Bottom Vertices

Select the control points that define the lower face of the chute.

Finish

Places the chute in the model.

Select Parent

Select the parent system for the chute in the Workspace Explorer.

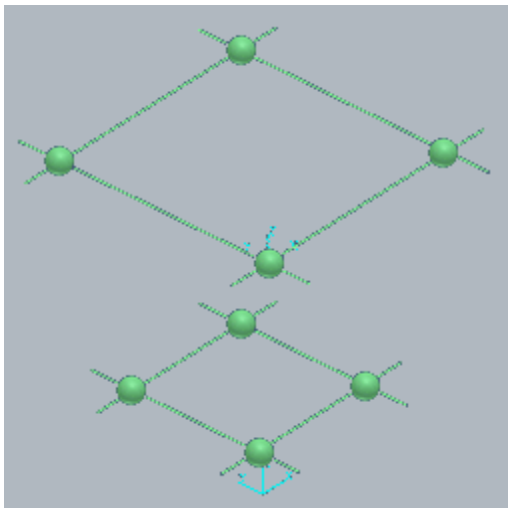
What do you want to do?

- *Create chute using control points in the same plane* (on page 128)
- *Create chute using control points on different planes* (on page 129)
- *Modify a chute by moving a control point* (on page 130)
- *Modify a chute by moving a grid plane* (on page 130)
- *Export chute plate geometry* (on page 130)

Create chute using control points in the same plane

1. Create grids on the needed planes and control points at the grid intersections. For more information, see *Create control points at grid intersections* (on page 123).

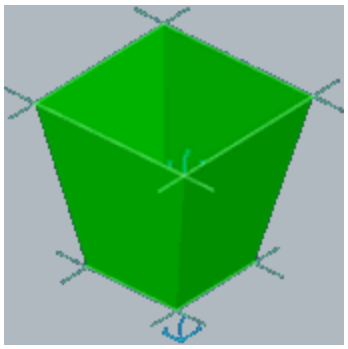
   Example:

2. In the Material Handling task, click **Create Chute** on the vertical toolbar.
3. Click **Properties**, and define the required plate properties. Click **OK**.
4. For **Select Top Vertices**, select the control points that define the top opening.

   ★ IMPORTANT Select the control points in either a clockwise or anti-clockwise direction, using the same direction for both the top and bottom openings.
5. Click **Select Bottom Vertices**.
6. Select the control points that define the bottom opening.
7. Click **Finish**.

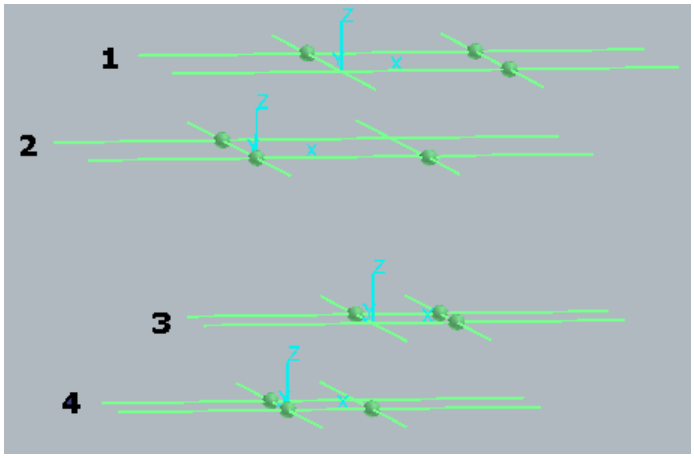
   The chute is created as a set of connected plate systems.

Create chute using control points on different planes

1. Create grids on the needed planes and control points at the grid intersections. At least one edge has control points in more than one plane. For more information, see *Create control points at grid intersections* (on page 123).

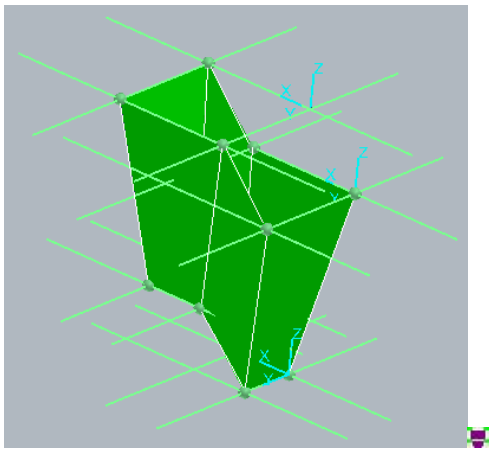
   In the following example, control points for the top edges are on planes 1 and 2. Control points for the bottom edges are on planes 3 and 4:



2. In the Material Handling task, click **Create Chute** on the vertical toolbar.
3. Click **Properties**, and define the required plate properties. Click **OK**.
4. For **Select Top Vertices**, select the control points on the planes that define the top opening.

   IMPORTANT Select the control points in either a clockwise or anti-clockwise direction, using the same direction for both the top and bottom openings.
5. Click **Select Bottom Vertices**.
6. Select the control points on the planes that define the bottom opening.
7. Click **Finish**.

   The chute is created as a set of connected plate systems.



Modify a chute by moving a control point

1. In the Material Handling task, select one or more control points in the graphic view.
2. Click **Properties** .

   *The **Control Point Properties** dialog box appears.*
3. If the control points are at a grid intersection, select **False** as the value for **Associativity** on the **General** tab.
4. Click **OK**.
5. In the graphic view, select the control points and move them to new locations.

   Chute plates are modified to the new control point locations.

Modify a chute by moving a grid plane

1. In the Grids task, click **Select** on the vertical toolbar.
2. Select **Elevation Plane** in the **Locate Filter** box.
3. Click an elevation plane having control points associated to a chute.

   *The **Place Grid Plane** ribbon appears.*
4. In the **Offset** box, type the required value and then press **Enter**.

   Control points on the grid plane move with the grid plane. Chute plates are modified to the new control point locations.

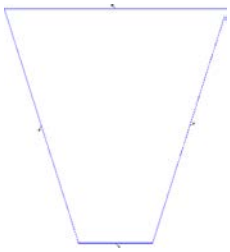
Export chute plate geometry

Export geometry of a chute plate to Sketch 2D. This enables you to use 2D automation tools for adding wear plates. The 2D geometry of a single chute plate is exported from the 3D model and used for placement of wear plates.

1. Click **Tasks** > **Molded Forms**.

   The Molded Forms task appears.
2. Click **Openings** on the vertical toolbar.
3. Select the chute plate system on which to place the wear plates.
4. Click **Opening Shape** and then click **Sketch 2D** on the ribbon.

   The Sketch 2D window displays with the 2D geometry of the chute plate.



5. Click **Edit** > **Select All**, and then click **Edit** > **Copy**.
6. Click **Cancel**.

   Smart 3D returns to the 3D graphic view.
7. Open Sketch 2D and create a new document.
8. Click **Edit** > **Paste** and then **Fit**  .

   The 2D geometry of the chute plate is visible in the document.
9. Click **File** > **Save**.

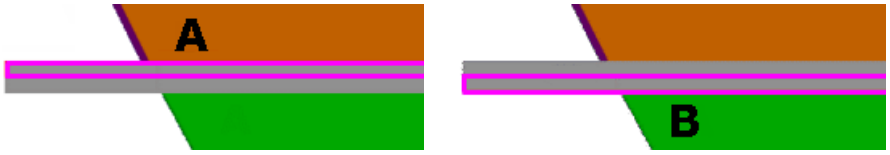
   A Sketch 2D .igr drawing file is saved in the selected location.

IMPORTANT The geometry for each chute plate system requiring wear plates must be on a separate sheet in one .igr file, or in separate .igr files.

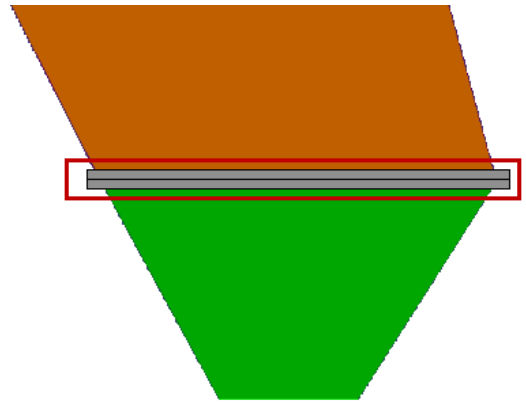
SECTION 15

Place Fastener Openings

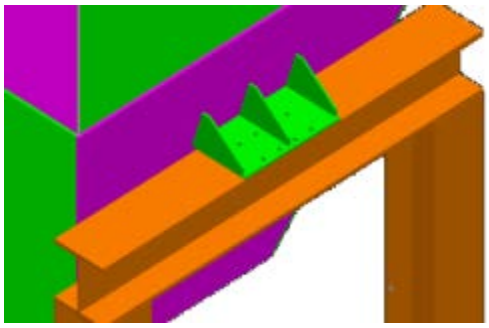
Creates openings for fasteners on the chute flanges. Chutes are typically manufactured as assemblies with flanges on the top (**A**) and bottom (**B**) portions of the chute, as shown in the examples below.



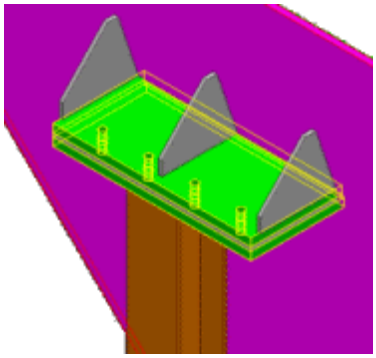
These assemblies are joined on site using fasteners that have been placed on the flanges. In the example below, the joined flanges of the chute assembly are shown in the red box.



Fasteners are also used to join support structure for a chute.



Fastener openings can be placed on up to four profile, members, or plates that are layered together, such as a series of plates, or two flanges with a plate between them. In the example below, three plates are shown.



You can place fastener openings on up to four profiles, edge reinforcements, members, and planar plates. On profiles and edge reinforcements, fastener openings are sketched features (similar to the sketched features in the Structural Detailing task). On plates, they are normal openings (similar to the openings created in the Molded Forms task). On members, openings are normal openings, similar to openings created in Structure.

Two placement modes are available: **Equal Spacing** and **Unequal Spacing**. Each mode requires that you define different types of input data. For more information about the placement modes and their respective input data requirements, see *Place Fastener Openings Dialog Box* (on page 138).

★ IMPORTANT

- Profiles and edge reinforcements must be detailed before placing fastener openings. Members and planar plates do not require detailing.
- Placement of fastener openings on planar plates is limited to planes that have been defined using the **Coincident** or **Offset from Plane** methods. For more information about the methods used to define planes, see *Plane Methods* in the *Molded Forms User's Guide*.

NOTES

- The diameter of each fastener opening created by the software is **(BoltDiameter + 2mm)**.
- Several generic types of fasteners are delivered in the Material Handling catalog. However, you can use the **HexagonalBolt** sheet in the **MaterialsHandlingEquipment.xls** workbook to customize the catalog to meet your required standards. For more information about customizing the catalog, see *Equipment Workbooks* in the *Material Handling Reference Data Guide*.
- Click **Preview** on the **Place Fastener Openings** dialog box to view the opening position before the software creates it. Click **Apply** or **OK** to create the openings, or click **Undo** to delete the opening place in the preview.

Reports

Use the **Tools** > **Run Reports** command to generate a stand-alone fastener report. The contents of the report are based on the openings created by **Place Fastener Openings** and includes information such as the diameter, quantity, grade, standards, and weight of the fasteners that you have placed in the model. For more information about reports, see *Run Report* in the *Common User's Guide*.

Place Fastener Openings Dialog Box (on page 138)

Select Equipment Dialog Box (on page 141)

What do you want to do?

- *Place fastener openings using equal spacing* (on page 134)
- *Place fastener openings using unequal spacing* (on page 135)
- *Calculate the fastener that meets your requirements* (on page 137)
- *Calculate the fastener that meets your requirements* (on page 137)
- *Generate a Fasteners report* (on page 138)

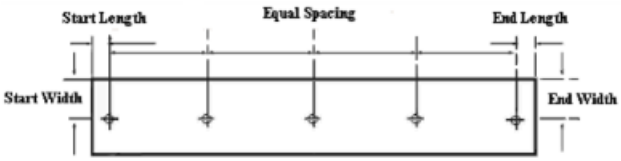
Place fastener openings using equal spacing

In **Equal Spacing** mode, the software creates equidistant fastener openings on the selected flange or plate. The distance between each flange is calculated based upon the location of the first fastener opening and the defined **Along Length** distances.

1. Click **Place Fastener Openings** on the vertical toolbar.

   *The **Place Fastener Openings** dialog box appears.*

   Place Fastener Openings Dialog Box (on page 138)
2. Specify the **Fastener Type**. You can use the following techniques:
   - *Calculate the fastener that meets your requirements* (on page 137)
   - *Select the fastener from the catalog* (on page 138)
3. In the **Place Fastener Openings** dialog box, define **Start** and **End** values for **Along Length** and **Along Width**.
4. Under **Mode**, select **Equal Spacing**.
5. Specify the number of fasteners to be placed in the **No. of Fasteners** box.
6. Optionally, click **Preview** to view a diagram that illustrates the placement of the openings.



7. In the graphic view, select two to four objects on which you want to place fastener openings. You can select plates, members, profiles and edge reinforcements.

   IMPORTANT
   - You can select any combination of objects, but only one member can be selected.

- Selections must be made using the graphic view. You cannot select using the **Workspace Explorer**.
- For profiles and edge reinforcements, you must select the detailed part.

8. Click **Preview**.

   TIP Click **Undo** to clear the preview, and then change any values on the dialog box. Reselect the objects on which you want to place fastener openings, and then click **Preview**.

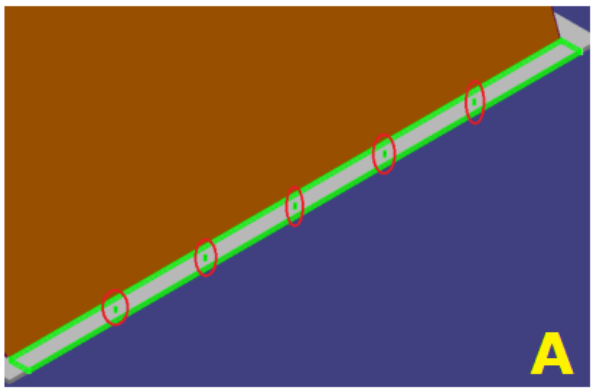
   The software displays a preview of the selected fastener openings based on the defined settings.

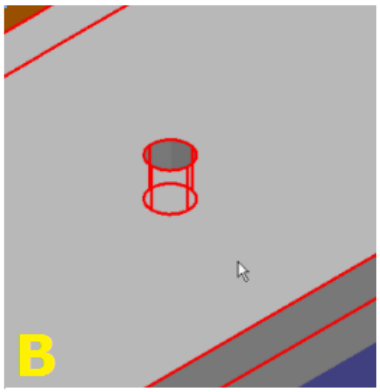
9. Click **OK**.

   NOTE Optionally, click **Apply**. After the software creates the required fastener openings, the dialog box remains open so that you can define settings to place additional fastener openings in the graphic view.

   *The software creates the required fastener openings based on the defined settings. During processing, the dialog box remains open and displays a progress bar. The message "Placing Fastener Openings" displays on the status bar. As the software determines the position of each opening on the flange, the view in the **Workspace Explorer** updates to show its placement. After all of the fastener openings have been placed, the software closes the dialog box.*

   *In the first example below, five fastener openings (circled in red) have been placed on the flange (**A**). The second example shows a zoomed in view of a fastener opening (**B**).*





Place fastener openings using unequal spacing

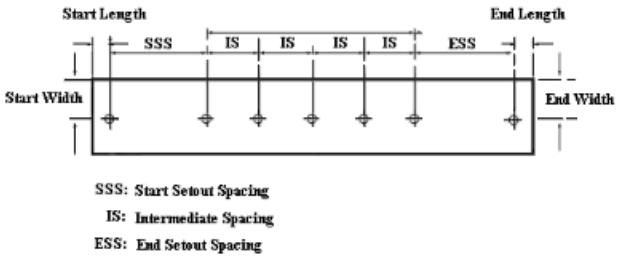
In **Unequal Spacing** mode, the number of openings placed on the selected flange or plate is determined by the values you specify for the **Along Length** distances and the defined spacing details.

1. Click **Place Fastener Openings** on the vertical toolbar.

   *The **Place Fastener Openings** dialog box appears.*

   Place Fastener Openings Dialog Box (on page 138)

2. Specify the **Fastener Type**. You can use the following techniques:
   - *Calculate the fastener that meets your requirements* (on page 137)
   - *Select the fastener from the catalog* (on page 138)
3. In the **Place Fastener Openings** dialog box, define **Start** and **End** values for **Along Length** and **Along Width**.
4. Under **Mode**, select **Unequal Spacing**.
5. Define values for **Start Setout Spacing**, **End Setout Spacing**, and **Intermediate Spacing**.
6. Optionally, click **Preview** to view a diagram that illustrates the placement of the openings.



7. In the graphic view, select two to four objects on which you want to place fastener openings. You can select plates, members, profiles, and edge reinforcements.

   IMPORTANT
   - You can select any combination of objects, but only one member can be selected.
   - Selections must be made using the graphic view. You cannot select using the **Workspace Explorer**.
   - For profiles and edge reinforcements, you must select the detailed part.
8. Click **Preview**.

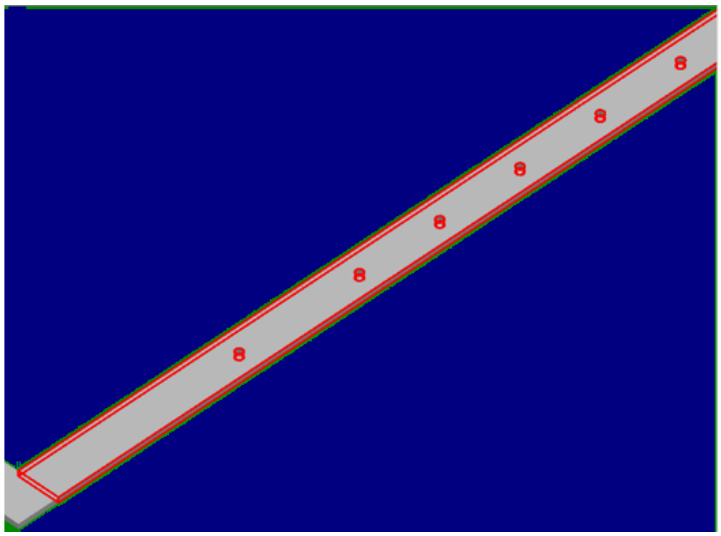
   TIP Click **Undo** to clear the preview, and then change any values on the dialog box. Reselect the objects on which you want to place fastener openings, and then click **Preview**.

   The software displays a preview of the selected fastener openings based on the defined settings.
9. Click **OK**.

NOTE Optionally, click **Apply**. After the software creates the required fastener openings, the dialog box remains open so that you can define settings to place additional fastener openings in the graphic view.

The software creates the required fastener openings based on the defined settings. During processing, the dialog box remains open and displays a progress bar. The message "Placing Fastener Openings" displays on the status bar. After all of the fastener openings have been placed, the software closes the dialog box.

The example below shows the fastener openings as placed on the flange.



IMPORTANT If you delete or modify the fastener opening on a stiffener or ER profile, then the corresponding fastener opening on the second profile also must be deleted or modified. This prevents any inaccuracies in the reports for the number of fasteners.

Calculate the fastener that meets your requirements

1. Click **Select Configuration**.

   *The **Fastener Details** dialog box displays.*

   Fastener Details Dialog Box (on page 140)

2. Select the **Fastener Type** from the list.

   *The software loads the available fastener sizes for that type into the **Fastener Size** list.*

3. Select the **Fastener Size** from the list.
4. Select the **Washer Configuration** from the list.
5. Click **Calculate**.

   The software calculates the length of fastener necessary to meet your requirements, and finds the closest match in the catalog.

6. Click **Apply**.

   *Smart 3D writes the value from the **Selected Standard Fastener** box to the **Fastener Type** box on the **Place Fastener Openings** dialog box.*

Select the fastener from the catalog

1. In the **Fastener Type** list, select **More**.

   *The **Select Equipment** dialog box appears.*

   Select Equipment Dialog Box (on page 141)

2. Expand the **Equipment** node, and double-click **Material Handling Equipment**.
3. Expand the **Fasteners** node, select the required fastener, and click **OK**.

Generate a Fasteners report

1. Click **Tools > Run Reports**.

   *The **Run Report** dialog box appears.*

2. On the **Catalog Reports** tab, expand the tree view, and select **Materials Handling**.
3. Select **Materials Handling_MHE_FastenerReports** in the list of available report templates.
4. Click **Run**.

   *The **Select Filter** dialog box appears.*

5. Select the appropriate filter, and click **OK**.

   *The **Select Filter** and **Run Report** dialog boxes close, and the report creation process begins. When processing completes, the software opens the report so that you can view its contents.*

Place Fastener Openings Dialog Box

Sets options that control how the software creates fastener openings in the model. Two placement modes are available: **Equal Spacing** and **Unequal Spacing**. The mode that you select determines which options must be defined for the software.

Fastener Type

Select the type of fastener you want to place in the model. The list displays the previous fastener that was selected. To view all of the fasteners that are in the catalog, click **More** to open the **Select Equipment** *dialog box* (on page 141).

Select Configuration

Displays the **Fastener Details** dialog box so that you can provide the information necessary to let the software determine the standard fasteners from the catalog that best meet your requirements. This option is only available after you select the object to which the fasteners are associated. For more information, see *Fastener Details Dialog Box* (on page 140).

Preview

Displays an image that illustrates fastener placement on the flange based on the selected placement mode and the corresponding distance or spacing values that you have defined.

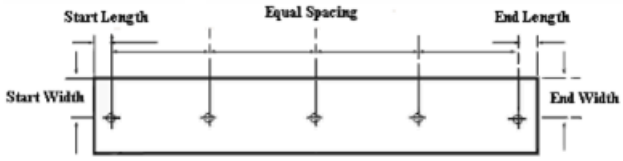
Distance from Edges

Along Length

Specifies the **Start** and **End** distances from the edge that fastener openings are to be placed along the length of the flange. These distances are represented as **Start Length** and **End Length** in the diagram below.

Along Width

Specifies the **Start** and **End** distances from the edge that fastener openings are to be placed along the width of the flange. These distances are represented as **Start Width** and **End Width** in the diagram below.



Placement Data

Mode

Equal Spacing

Places equidistant fasteners on the flange so that they are equidistant from one another. This option requires that you also specify a value for **No. of Fasteners**. The distance between each flange is calculated based upon the location of the first fastener opening and the defined **Along Length** distances.

Unequal Spacing

Places fastener openings on the flange as required based upon the spacing details that you define. Selecting this option also requires that you define values for **Start Setup Spacing**, **End Setup Spacing**, and **Intermediate Spacing**.

No. of Fasteners

Specifies the number of fasteners to be placed on the flange. This option is available only when **Equal Spacing** is the selected placement mode.

Spacing Details

Start Setout Spacing

Specifies the distance between the first and second fastener openings on the flange. This distance is represented by **SSS** in the diagram below. This option is available only when **Unequal Spacing** is the selected placement mode.

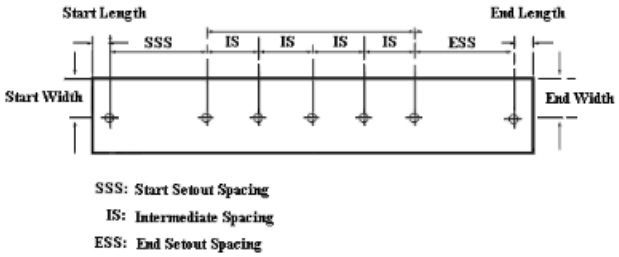
End Setout Spacing

Specifies the distance between the last two fastener openings placed on the flange. This distance is represented by **ESS** in the diagram below. This option is available only when **Unequal Spacing** is the selected placement mode.

Intermediate Spacing

Specifies the distance between the intermediate fastener openings. That is, the fastener openings that are placed between the first and last fastener on the flange. This distance is

represented by **IS** in the diagram below. This option is available only when **Unequal Spacing** is the selected placement mode.



Preview

Displays a preview of the selected fastener openings based on the defined settings.

Undo

Removes the last **Preview** results.

Fastener Details Dialog Box

Sets options to select the fastener from the catalog that best meets your criteria.

Fastener Type

Specifies the type of fastener (**Hexagonal Headed** or **Counter Sunk**).

Fastener Size

Specifies the size of fastener. The list contains the options from the catalog that correspond to the **Fastener Type**.

Washer Configuration

Specifies the washer configuration associated with the fastener.

Calculate

Determines the **Calculated Length**, **Selected Length**, and **Selected Standard Fastener** values.

Calculated Length

Displays the minimum fastener length that meets your requirements.

Selected Length

Displays the fastener length from the catalog that most closely meets your requirements.

Selected Standard Fastener

Displays the catalog name of the fastener that most closely meets your requirements.

Apply

Posts the **Selected Standard Fastener** value in the **Fastener Type** box on the *Place Fastener Openings Dialog Box* (on page 138).

Cancel

Closes the dialog box without posting any values in the *Place Fastener Openings Dialog Box* (on page 138).

Select Equipment Dialog Box

Specifies the equipment type needed for placement. This dialog box appears automatically when you select **More** in the **Fastener Type** list on the **Place Fastener Openings** dialog box. By browsing through the part hierarchy, you can find any type of fastener in the catalog. After you select a fastener and click **OK**, the software returns you to the **Place Fastener Openings** dialog box so that you can define the settings required to place the fastener openings.

 **Properties**

Displays the equipment properties as defined in the catalog.

 **Preview**

Displays a bitmap symbol of the selected equipment. The image file must be assigned to the equipment in the catalog reference data.

 **List View**

Sets the dialog box to display equipment in a list view.

 **Grid View**

Sets the dialog box to display equipment in a spreadsheet-style grid view.

 **Back**

Returns you to the previously selected equipment part or node. Use this command to navigate through the equipment hierarchy to the specific part you need.

 **Forward**

Sends you to the last selected equipment part or node that you moved away from by using the **Back** button. Use this command to navigate through the equipment hierarchy to the specific part you need.

 **Up One Level**

Brings up the next highest level of the equipment catalog hierarchy. Use this command to navigate through the hierarchy to the specific part you need.

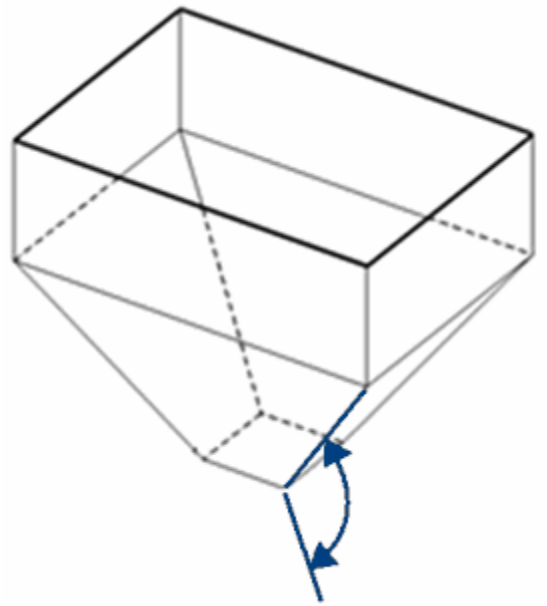
Address

Specifies your exact location within the displayed hierarchy.

SECTION 16

Measure Valley Angle

Calculates the angle at the intersection of any two selected planes or plates. The valley angle is typically measured at the corner created by two inclined plates on a chute.



NOTE Valley angle cannot be calculated on a horizontal plate or plane.

Valley Angle Ribbon

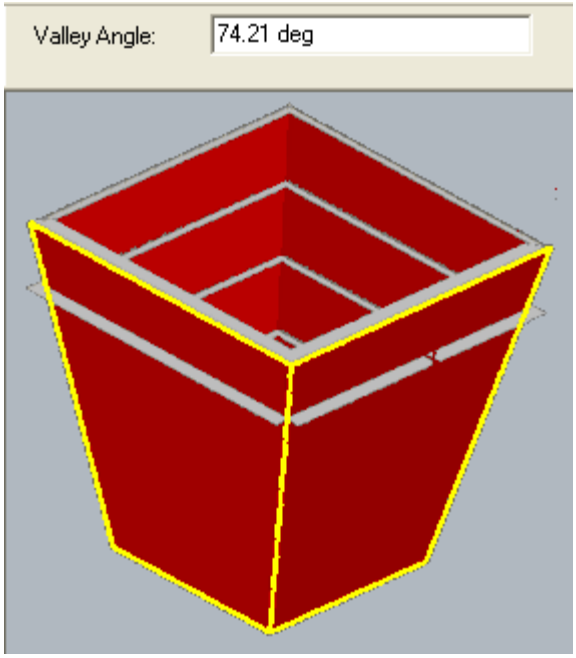
Valley Angle

Displays the valley angle at the intersection of the selected plates or planes.

Measure the valley angle between plates on a chute

1. In the Material Handling task, click **Measure Valley Angle** .
2. In the graphic view, select a plate system on a chute.
3. Select a second plate system on the chute.

*The value of the valley angle at the corner of the two plates appears in **Valley Angle**.*



SECTION 17

Create Volume View

 Creates a view from a drawing volume. You can control how objects display in the workspace using the same volume boundaries that define your volume drawings. The command does the following:

- Uses a drawing volume to clip objects in the workspace.
- Defines an orientation for the view.
- Defines a name for the view and adds it to the **Named Views**  list.

Before using this command, you must:

- Create a volume drawing component with one view in the Drawings and Reports task.
- Place a drawing volume in the Space Management task.

Create Volume View Ribbon

Finish

Places the volume view in the model.

Name Suffix

Type a name for the volume view. The name will be appended to the selected volume name.

View Direction

Select the viewing direction of the volume view. The graphic view changes to this direction when the volume view is selected. Select **Front**, **Top**, **Right**, or **Isometric**.

What do you want to do?

- *Create a volume drawing component* (on page 145)
- *Create a volume* (on page 146)
- *Create a volume view* (on page 146)
- *Display a volume view* (on page 147)

Create a volume drawing component

1. Click **Tasks** > **Drawings and Reports**.
2. In the **Management Console**, right-click the Model root in the hierarchy, and select **New > More**.

   *The **Add Component** dialog box appears.*
3. On the **General** tab, select **Folder**, and click **OK**.

   New Folder *appears in the Console.*
4. Right-click **New Folder**, and select **Rename**.
5. Type a name for the folder, such as *Volume Views*.
6. Right-click the folder, and click **New > More**.

   *The **Add Component** dialog box appears.*
7. On the **General** tab, select **Volume drawings**, and click **OK**.

   *The **New Volume Drawings** component appears under the folder in the Console hierarchy and in the Detail View.*
8. Right-click **New Volume Drawings**, and select **Edit Template**.

   *The **Select Template** dialog box appears.*
9. Select a template, and click **OK**.

   The template opens in the Sketch 2D Drawing Editor.
10. Click **Place Drawing View**.
11. Click in the drawing, hold the mouse button down while dragging the mouse diagonally, and release.

    A view is created on the drawing.

    *The **Drawing View Properties** dialog box appears.*
12. On the **Drawing View Properties** dialog box, enter the needed information on the **View** tab in the **Name**, **Description**, **Style**, **Orientation**, and **Scale** boxes.
13. Click **OK**.
14. Click **File** > **Save,** and close Sketch 2D Drawing Editor.

NOTE For more information, see *Volume Drawings* in *Orthographic Drawings User's Guide.*

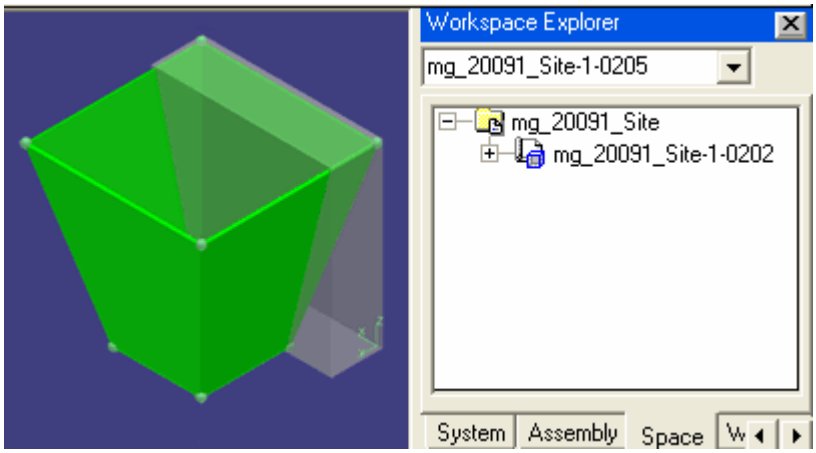
Create a volume

1. In the Space Management task, click **Place Drawing Volume by Selection** on the vertical toolbar.
2. In the graphic view or **Workspace Explorer**, select one or more objects to include in the volume.

   TIP Select objects that are at the outer boundaries of the needed volume.
3. For **Drawing Type**, click **More**, select a volume drawing component, and click **OK**.
4. For **Space Folder**, click **More**, select the model database, and click **OK**.

   TIP Use **Create Space Folder** to organize volumes on the **Space** tab. For more information, see *Create Space Folder* in *Space Management User's Guide*.
5. Click **Finish**.

   *The volume appears in the graphic view and on the **Space** tab of the **Workspace Explorer**.*



NOTE Other drawing volume commands can be used to create a volume. For more information, see *Drawing Volumes* in *Space Management User's Guide*.

Create a volume view

1. In the Material Handling task, click **Create Volume View** on the vertical toolbar.
2. In the graphic view, select a drawing volume.

   NOTE You can also select a drawing volume on the **Space** tab of the **Workspace Explorer**.
3. For **Name Suffix**, type text to append to the selected volume name.
4. For **View Direction**, select a value.
5. Click **Finish**.

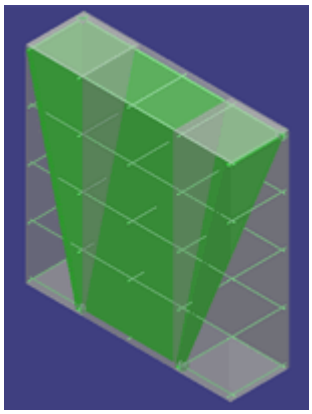
Display a volume view

1. Click **View** > **Named Views** or **Named Views** on the horizontal toolbar.

   *The **Named Views** dialog box appears. It displays all of named views created, including volume views.*

2. Select the needed volume view and click **OK**.

   The selected volume view displays in its defined orientation. Objects in the model are clipped to the volume view boundaries.



NOTE For more information, see *Named Views* in the *Common User's Guide*.

SECTION 18

Modify Thickness Direction

Changes the plate thickness direction of a chute, and aligns the primary profile orientation of the root plate with the thickness direction, allowing you to place stiffeners on the outside of the chute. This command is only applicable for general plates.

NOTE For **Planar Plate System**, **Linear Extruded Plate System**, **Non-Linear Extruded Plate System**, **Revolved Plate System**, **Bracket Plate System**, **Child Plate System**, and **Ruled Plate System** plate systems, **General Plate** is the default plate type with S3D-Materials Handling Edition.

Material Thickness Direction Ribbon

Select a point

Places a point in the workspace. This command is active by default.

Select Plates / Parent System

Selects the general plates or parent system of a chute.

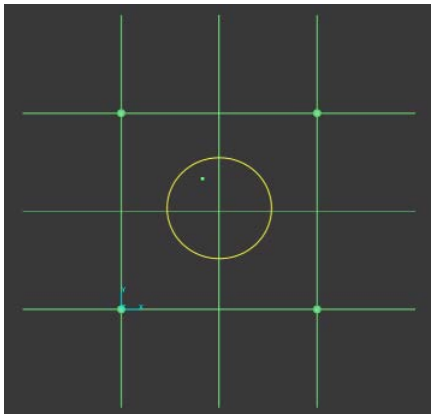
Finish

Completes the modify thickness direction action.

Modify thickness direction

1. Locate the appropriate chute.
2. Click **Tools** > **Modify Thickness Direction Command**.
3. Click **Named Views**, and then select **Top** to change the perspective angle of the view, so that you are looking down at the model.
4. Click near the center of the chute.

   The software places a green point as shown in the following graphic.



5. In the **Workspace Explorer**, select the corresponding plate systems of the chute.
6. Click **Finish**.

   *On the **Molded Conventions** tab (**Chute Plate System Properties** dialog box), you can observe that the plate **Thickness Direction** and **Primary Orientation** have been modified to **With Normal** or **Opposite Normal**. For more information see,* Molded Conventions Tab (Chute Plate System Properties Dialog Box) *(on page 150).*

NOTES

- This command is applicable only for general plates.
- The change in thickness direction of the plate is outwards from the center of the chute.

Chute Plate System Properties Dialog Box

Specifies the properties for the chute plate system that you are editing.

Topics

Main Tab (Chute Plate System Properties Dialog Box)

Specifies the general properties of the chute plate system.

Name

Specify the name of the plate system. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the Global Workshare Guide.

Rule

Select the naming rule to use to name the plate system.

- Select **StdPlateSystemNamingRule** to use the following syntax: *<Reference Plane Name>-<Index Number><Plate Type>-<Workshare Location ID>*.
- Select **User Defined** if you want to name the plate yourself by using the appropriate box.

Type

Select the type of plate system that you are placing. Chute plate systems have only one type: **General Plate**.

Naming Category

Select a category for the plate system. Categories specify the role of the plate system in the plant. The category is also used by the naming rule to name the plate part that is a child to the plate system.

Parent System

Select a parent system for the plate system. You can define parent systems in the Systems and Specifications task. When the plate system is created, the property values for the parent system are used for the initial plate system property values. When a parent property value changes, the corresponding child property value is updated.

Surface Geometry Type

This is a Molded Forms property and is not used by the Material Handling task.

Specification

Select the structural specification for the plate system.

Description

Type a description for the plate system. This property is optional.

Continuity

This is a Molded Forms property and is not used by the Material Handling task.

Priority

This is a Molded Forms property and is not used by the Material Handling task.

Tightness

This is a Molded Forms property and is not used by the Material Handling task.

Material Tab (Chute Plate System Properties Dialog Box)

Specifies the material properties for the chute plate system that you are editing.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

Specifies the object material grade, such as **A36** or **A529**.

Thickness

Specifies the material thickness for the plate system.

Molded Conventions Tab (Chute Plate System Properties Dialog Box)

Specifies the default plate and profile orientation rules for the chute plate system.

Plate System Type

Displays the plate system type that you specified on the **Main** Tab. Chute plate systems have only one type: **General Plate**.

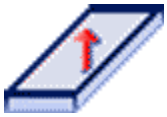
Plates

Plate Thickness Direction

Select the direction from the plate system molded surface in which the plate thickness should be applied. The default thickness direction is defined by the molded conventions in the Catalog.

- **With Normal** - The thickness direction is in the same direction as the normal vector of the plate system molded surface.

- **Opposite Normal** - The thickness direction is in the opposite direction from the normal vector of the plate system molded surface.



- **Centered** - The thickness is on both sides of the plate system molded surface.

Offset

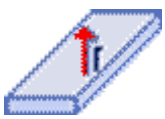
Specify the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles

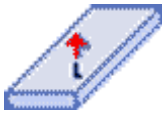
Primary Orientation or Profile Orientation

Set the primary orientation for the profile web. The primary orientation is dependent on the **Plate Thickness Direction** value.

- **With Normal** - The profile web is on the plate side that is in the same direction as the normal vector of the plate system molded surface.



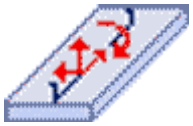
- **Opposite Normal** - The profile web is on the plate side that is in the opposite direction from the normal vector of the plate system molded surface.



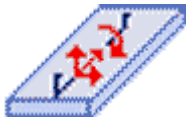
Secondary Orientation

Set the secondary orientation for the profile flange.

- **Left** - The profile flange is on the left when looking in the direction of the plate normal vector.



- **Right** - The profile flange is on the right when looking in the direction of the plate normal vector.



SECTION 19

Export DSTV

The **File** > **Export** > **DSTV** command creates and exports a DSTV file from data in a manufacturing profile XML file. The XML file is created in the Structural Manufacturing task. DSTV is a CNC file format used as an input to CNC machines for profile manufacturing.

Before using this command, you must:

- Add all needed profile systems, such as adding profiles systems to plate systems created with **Create Chute**  . For more information, see the *Molded Forms User's Guide*.
- Detail the needed structure and add any needed detailing features. For more information, see the *Structural Detailing User's Guide*.
- Create manufacturing profiles. For more information, see *Manufacturing Profile* in the *Structural Manufacturing User's Guide*.

Export DSTV File Dialog Box (on page 154)

What do you want to do?

- *Create a manufacturing XML file* (on page 153)
- *Export a DSTV file from a Manufacturing XML File* (on page 154)

Create a manufacturing XML file

1. In the Structural Manufacturing task, select one or more manufacturing profiles in the graphic view or the **Workspace Explorer**.
2. Click **Part Monitor**  on the vertical toolbar.
3. In the **Viewer** box, select **Text**.
4. In the **Output Format** box, select **Members SMS_PROFILES**.
5. Click **View**.

   *The **Preview XML Data** dialog box appears, showing XML data for the profile.*
6. In the dialog box, click **File** > **Save**.
7. Save the XML file to the needed location.

NOTE For more information, see *Part Monitor* in the *Structural Manufacturing User's Guide*.

Export a DSTV file from a manufacturing XML File

1. Click **File** > **Export** > **DSTV**.

   *The **Export DSTV File** dialog box appears.*
2. In the **Description** box, type a description of the XML file.
3. In the **Author** box, type your name.
4. In the **Organization** box, type your company name.
5. In the **XML File** box, specify the location and name of a manufacturing profile XML file for the export operation. Type, or click **Browse**.
6. In the **DSTV File Prefix** box, specify the location and name of the DSTV file. Type, or click **Browse**.
7. In the **Log File** box, specify the location and name for a log file created during the export. Type, or click **Browse**.
8. Click **Apply**.

   *The software creates the DSTV file. When processing completes, **View NC File** is available.*
9. Click **View Log**.

   The log file opens.
10. Click **View NC File** to see the DSTV file.

    The folder specified for the DSTV file opens.

NOTE In order to view the log and the NC file by clicking **View Log** and **View NC File**, you must click **Apply**. If you click **OK**, the dialog box is closed when processing is complete.

Export DSTV File Dialog Box

Specifies settings for the DSTV file.

Filter

Specify the workspace filter. Manufacturing objects included in the filter are selected. *This option is currently not available*.

Description

Type a description. This description is included in the DSTV file.

Author

Type a name to include in the DSTV file.

Organization

Type the company or organization name to include in the DSTV file.

XML File

Specify the XML file to export. Click **Browse** to browse for the XML file.

DSTV File Prefix

Specify the location and name for the DSTV file. The software appends a numeric counter (_1, _2) to each file name. Type, or click **Browse**.

Log File

Specify the location and name for the log file. The log file tracks the export operation. Type, or click **Browse**.

OK

Starts the export process. The **Export DSTV File** dialog box is closed when processing is complete.

Cancel

Exits the **Export DSTV File** dialog box.

Apply

Starts the export process. The **Export DSTV File** dialog box remains open when processing is complete.

View Log

Opens the export log file. Available if **Apply** is clicked.

View NC File

Opens the folder where the DSTV file is created. This button is available after the DSTV file is successfully exported. Available if **Apply** is clicked.

SECTION 20

Profile Auto Bound

Creates mutual bounding between profile edge reinforcements on chute plates. This command is on the **Tools** menu.

NOTES

- Chute plates can be mutually bounded or single-bounded.
- The distance between the stiffeners or edge reinforcements must be between 0 and 0.01 meters.

Profile Auto Bound Ribbon

Controls parameters for mutual bounding between stiffeners profiles and profile edge reinforcements.

Select Root Plate Systems

Select the chute plates connected to the stiffeners profiles or profile edge reinforcements. The chute plates can be mutually bounded or single-bounded, and the distance between the stiffeners or edge reinforcements must be between 0 and 0.01 meters.

Finish

Complete the bounding.

Automatically bound stiffeners profiles or profile edge reinforcements on chute plates

1. Click **Tools > Profile Auto Bound**.

   *The **Profile Auto Bound** ribbon displays.*

2. Click **Select Root Plate Systems** .

3. Select the chute plates.

   The software highlights the plates as you select them.

4. Click **Finish**.

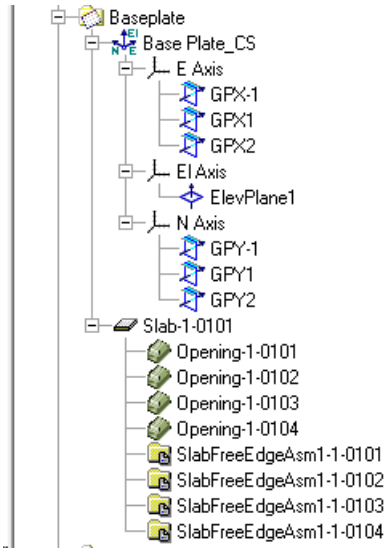
   The software automatically bounds the stiffeners profiles or profile edge reinforcements that are present on the mutually bounded chute plates.

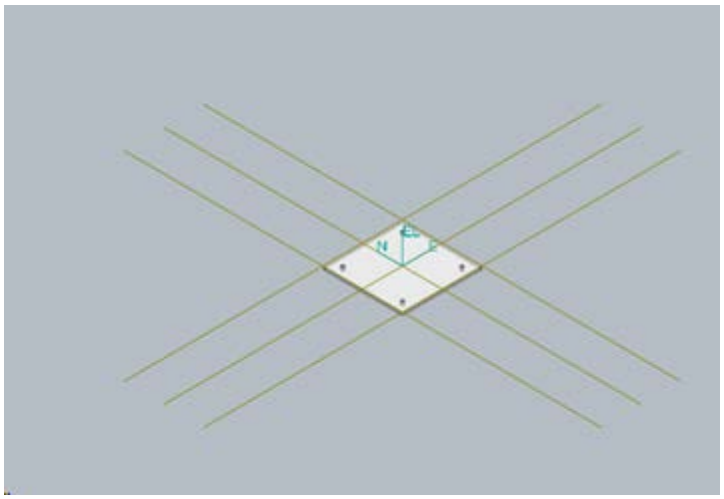
SECTION 21

Place Base Plate

Places base plates from the catalog on specified columns.

NOTE Create a generic system with the name to use for the base plate. Then, create a grid system with one elevation plane and the required number of X and Y planes to place the slab and any required openings. Create the base plates using **Place Slabs** in Structure. Change the material type to steel. For more information, see *Place Slabs* in the *Structure User's Guide*. After you model the base plates, including any necessary openings, copy the base plate to the catalog using the generic system as the root. For more information, see *Copy to Catalog* in the *Common User's Guide*.





Place Base Plate Ribbon

Select Structural Members

Specifies the structural members under which to place the base plates. You can select the members in the model or in the **Workspace Explorer**.

Select Base Plate

Specifies the base plate to place. Click **More** to browse through the catalog hierarchy.

Finish

Places the placement point of the baseplate at the bottom center of each member.

Place a base plate

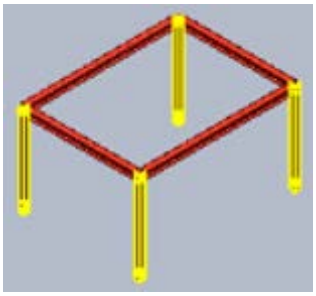
1. Click **Tools > Place Base Plate Command**.

   *The **Place Base Plate** ribbon displays.*

2. Select the structural members under which to place the base plates.

   TIP You can select the members in the model or the **Workspace Explorer**.

   The selected members highlight.



3. Select **More** from the **Select Base Plate** list.

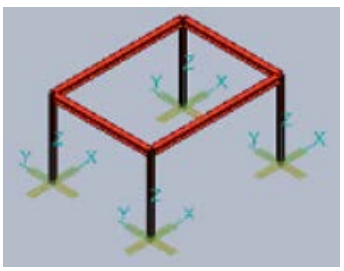
   *The **Select Equipment** dialog box displays.*

4. Browse through the hierarchy to select the base plate from the catalog.
5. Click **OK** on the **Select Equipment** dialog box.

   *The selected base plate name displays in the **Select Base Plate** box.*

6. Click **Finish** on the **Place Base Plate** ribbon.

   Smart 3D places the placement point of the base plate at the bottom center of each member.



SECTION 22

2D Automation in Sketch 2D

2D automation commands are used in Sketch 2D to place belt-related equipment such as pulleys, idlers, trusses, trestles, and control points.

NOTE 2D automation commands are delivered as custom commands in Smart 3D Material Handling and are added to **Sketch 2D**. The commands are available when **Sketch 2D** is clicked on the **Place Belt Components** ribbon, the **Place Trestle** ribbon, or the **Place Chute Shapes** ribbon. For more information, see *Place Belt Components* (on page 71), *Place Trestle* (on page 78), and *Place 2D Chute Shapes (Sketch 2D - Custom Command)* (on page 174).

Sketch 2D automation contains the following custom commands:

Place Belt Components places equipment symbols, such as pulleys, idlers, trusses, and control points on belt profile geometry and exports the symbols to the Material Handling 3D model. For more information, see *Place Belt Components* (on page 71).

Belt Correction Tool checks for any faults in a belt profile. You can check for multiple connections, non-belt segments, and missing continuity and constraints in belt segments. For more information, see *Belt Correction Tool (Sketch 2D - Custom Command)* (on page 170)

Place 2D Chute Shapes places chute shape symbols on a belt profile in Sketch 2D, and exports the symbols to the Material Handling 3D model. For more information, see *Place 2D Chute Shapes (Sketch 2D - Custom Command)* (on page 174).

Create Trajectory places trajectory paths along the material flow. The software adjusts the trajectory shape based on the conveyor type in the **Sketch 2D** environment. This command is useful for laying out the conveyor and using the **Place 2D Chute Shapes** command. For more information, see *Create Trajectory (Sketch 2D - Custom Command)* (on page 175).

Place 2D Trestle places trestle symbols on belt profile geometry and exports the symbols to the Material Handling 3D model. For more information, see *Place 2D Trestle (Sketch 2D - Custom Command)* (on page 176)

Idler Packing Thickness calculates packing thickness and adds additional packing plates between idlers and their associated truss. For more information, see *Idler Packing Thickness (Sketch 2D - Custom Command)* (on page 180).

Custom command behavior

Access to Sketch 2D commands is environment-specific, which means that only relevant Sketch 2D commands are accessible when the corresponding 3D environment command is active. For example, when you use the **Place Chute Shapes** command and then enter Sketch 2D, the software allows you to access the only relevant Sketch 2D commands: **Create Trajectory** and **Place 2D Chute Shapes**. Access to other Sketch 2D commands (**Belt Correction Tool**, **Place Belt Components**, and **Place 2D Trestle**) is restricted. When you try to use a restricted

command, the software displays a message stating that you can only access the command when the relevant Smart 3D command is active. The table below lists the Smart 3D commands and the relevant Sketch 2D command for each:

Smart 3D Command	Relevant Sketch 2D Commands
Place Conveyor Belt	Belt Correction Tool
Place Chute Shapes	Create Trajectory Place 2D Chute Shapes
Place Belt Components	Place Belt Components Idler Packing Thickness
Place Trestle	Place 2D Trestle

Data layer

Use the data layer to group related sets of symbols. Symbols representing 3D equipment, trusses, control points, or trestles can be placed on a data layer. The data is associated with a system in the 3D model. You can change the data layer at any time.

★ IMPORTANT Each data layer has a different parent system and should only contain data from one 2D automation command. For example, do no place idlers and trusses on the same layer.

Symbol folder

When running any of the commands for the first time, you need to browse to the folder containing the 2D symbols, and associated command configuration (.LST) files. Material Handling symbols are delivered in the Smart 3D SharedContent folder.

Placement point selection

Place Belt Components positions symbols along the belt profile. Before the first symbol is placed, however, an initial point on the belt profile needs to be selected. This point on the belt is the starting reference point. This is only true for belt components.

Equipment orientation

By default, equipment is placed normal to the belt. If a different orientation is needed, it can be specified via the **Orientation** option. This selection is only available when placing equipment type components.

Add custom commands to the Sketch 2D toolbar

Add a specific custom command to the toolbar

1. In the **Sketch 2D** window, right click on the vertical toolbar and select **Toolbars**.

   *The **Toolbars** dialog box appears.*

2. Click **Customize**.

   *The **Customize** dialog box appears.*

3. In the **Categories** box, scroll down and select **Custom Commands**.

4. In the **Custom Commands** box, click **Browse**.

   *The **Select Macro Directory** dialog box appears.*

5. In the **Folders** box, navigate to the *[Product Folder]*\MaterialsHandling\Client\Bin folder.

   *OCX or DLL command files in the folder are visible, but inactive, in the **Custom Commands** box.*

6. Click **OK**.

   *The OCX or DLL command files appear in the **Custom Commands** box of the **Customize** dialog box.*

7. Drag the needed command file from the **Custom Commands** box onto a blank area in the toolbar region of the main **Sketch 2D** window.

   The 2D custom commands delivered in Smart 3D Material Handling are:

   - MH2DFixBeltProfile.ocx. For more information, see *Belt Correction Tool (Sketch 2D - Custom Command)* (on page 170).
   - MHCreateTrajectory.ocx. For more information, see *Create Trajectory (Sketch 2D - Custom Command)* (on page 175).
   - MHPackingThickness.ocx. For more information, see *Idler Packing Thickness (Sketch 2D - Custom Command)* (on page 180).
   - MHPlace2DEquip.ocx. For more information, see *Place Belt Components (Sketch 2D - Custom Command)* (on page 162).
   - MHPlace2DShapes.ocx. For more information, see *Place 2D Chute Shapes (Sketch 2D - Custom Command)* (on page 174).
   - MHPlace2DTrestle.ocx. For more information, see *Place 2D Trestle (Sketch 2D - Custom Command)* (on page 176).

   A new toolbar window with the command icons appears.

8. Click **Close**.

Add all custom commands to the toolbar

1. In the **Sketch 2D** window, click **Tools > Custom Commands**.

   *The **Custom Command** dialog box displays.*

2. Navigate to the *[Product Folder]*\MaterialsHandling\Client\Bin folder in the **Look in** list.

3. Select the **MHPlace2DEquip.ocx** file.
4. Click **Open**.

   The software displays a toolbar that includes all of the custom commands.

Place Belt Components (Sketch 2D - Custom Command)

Places equipment symbols, such as *pulleys, idlers, trusses, and control points*, as well as module and miscellaneous symbols, on belt profile geometry in **Sketch 2D**. These symbols are then placed in the 3D environment.

Place Belt Components Dialog Box (Sketch 2D Custom Command) (on page 165)

Layer Properties Dialog Box (on page 169)

What do you want to do?

- *Place equipment symbols in 2D* (on page 162)
- *Place module symbols in 2D* (on page 163)
- *Place miscellaneous symbols in 2D* (on page 164)
- *Change equipment orientation* (on page 165)

Place equipment symbols in 2D

1. In **Sketch 2D,** click **Place Belt Components** .

   *The **Place Belt Components** dialog box appears.*
2. In **Select Belt Component,** select **Equipment**.
3. In **Symbol Directory,** type the full path location or browse to the needed Symbol Share folder.
4. Click the **Place Along** tab.
5. In the **Part Class** box, select the type of idler to be placed.

   *The **Part Name** box is populated with the parts of that class.*
6. In **Part Name**, select the idler part to place.
7. In **Count**, type the number of idler parts to be placed.
8. In **Offset**, type the offset from the start point for the position of the first part.
9. In **Spacing**, type the spacing between parts when **Count** is greater than 1.
10. The **Orientation** box is set to **Normal to Belt** by default. If not, click **Change** and select a new value.
11. In the **Selection Mode** box, select a method for creating the start point on the belt profile.
12. In the **Direction of Placement** box, select a direction value.

13. Based on the specified **Selection Mode**, select a start point on the belt profile in the graphic view of the drawing.

    *After a valid selection is made, the **Go** button is enabled.*

14. Click **Go** to place symbols for the idler parts.

NOTES

- Repeat these steps as many times as required to populate the belt with symbols for the selected part.
- Subsequent parts are placed relative to where the last idler was placed.
- On subsequent executions, if **Offset** is 0, the value of **Spacing** is automatically used as the first offset.
- When you place an equipment and then modify the geometry of the belt profile, the equipment gets distorted. To correct the distortion use the **Sync3D-2D** feature to realign the components in Sketch 2D according to their positions in 3D environment.

Place module symbols in 2D

1. In **Sketch 2D**, click **Place Belt Components** .

   *The **Place Belt Components** dialog box appears.*

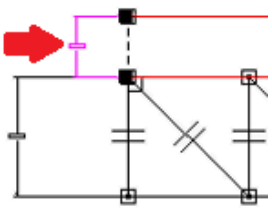
2. Set the **Select Belt Component** to **Modules**.
3. In **Symbol Directory,** type the full path location or browse to the needed Symbol Share folder.
4. Click the **Place Along** tab.
5. In the **Part Class** box, select the type of module to be placed.

   *The **Part Name** box is populated with the parts of that class.*

6. In **Part Name**, select the module part to place, such as a truss created using the **Truss Wizard** (on page 59).
7. In **Count**, type the number of module parts to be placed.
8. In **Offset**, type the offset from the start point for the position of the first part.
9. In **Spacing**, type the spacing between parts when **Count** is greater than 1.
10. In **Idler Height**, type in the appropriate height.

    *The software applies the **Idler Height** value to the 2D symbol.*

    TIP The idler height displays in the 2D symbol as a dimension between the top of the truss and a reference line. If you change this dimension, the idler height value changes accordingly.



The **Idler Height** value is stored on the symbol. To add this value to an existing symbol, add the **IdlerHeight** value on the **Parameters** tab of the **Symbol Properties** dialog box.

11. In the **Selection Mode** box, select a method for creating the start point on the belt profile.
12. In the **Direction of Placement** box, select a direction value.
13. Based on the specified **Selection Mode**, select a start point on the belt profile in the graphic view of the drawing.

    *After a valid selection is made, the **Go** button is enabled.*
14. Click **Go** to place symbols for the module parts.

NOTES

- Repeat these steps as many times as required to populate the belt with symbols for the selected part.
- Subsequent parts are placed relative to where the last module was placed.
- On subsequent executions, if **Offset** is **0**, the value of **Spacing** is automatically used as the first offset.

Place miscellaneous symbols in 2D

1. In **Sketch 2D**, click **Place Belt Components**  .

   *The **Place Belt Components** dialog box appears.*
2. Set the **Select Belt Component** to **Miscellaneous**.
3. In the **Symbol Directory**, type the full path location or browse the needed Symbol Share folder.
4. Click the **Place** tab.
5. In **Part Class** box, select **Object**.
6. In **Part Name**, select **Control Point**.
7. In **Type**, select **Control Point**.
8. In **Subtype**, select type of control point to be placed.
9. In **Name**, type in a name for the component.

   NOTE This is the name assigned to the control point created in the 3D model. If the **Name** is blank, the naming rule in the 3D model assigns a name.
10. Left-click to place the control points in Sketch 2D.

Change equipment orientation

1. In **Sketch 2D**, click **Place Belt Components**  .

   *The **Place Belt Components** dialog box appears.*
2. Click the **Place Along** tab.
3. Click **Change**.

   *The **Orientation** box is enabled.*
4. Select the appropriate orientation.
5. Click **Close**.

Place Belt Components Dialog Box (Sketch 2D Custom Command)

Sets options for placing equipment symbols, such as pulleys, idlers, trusses, and control points on belt profile geometry in **Sketch 2D**.

Configure Tab

Defines configuration information for the equipment symbols.

Place Tab

Options update relative to the component set on the **Configure** tab.

Close

Closes the dialog box.

Topics

Configure Tab (Sketch 2D Place Belt Components Dialog Box)

The **Configure** tab defines configuration information for the symbols.

Select Belt Component

Select a component type: **Equipment**, **Modules,** or **Misc**.

Symbol Directory

Type the full path location to the symbol share folder, or browse to select a symbol share.

Layer Name

Specify the required layer. Type a name in the box, or select a name in the list. To create a new layer, click **New Layer**.

New Layer

Opens the **Layer Properties** *dialog box* (on page 169) so that you can create a new layer. Layers created using **New Layer** are also visible in the **Layer Display** dialog box, which is accessible using the **Tools > Layers** command in Sketch 2D. For more information, see the Sketch 2D Help.

Properties

Opens the **Layer Properties** dialog box so that you can specify the name of the parent system.

Idler Height

Displays the offset distance between the 2D module and the belt profile. The software uses this distance to verify that idlers are appropriately supported on the module.

Change

Prompts you to select the modules to update. Select the modules in Sketch 2D.

Update

Changes the idler height value for all of the selected modules to the value displayed in the **Idler Height** box.

NOTES

- The components are displayed under the parent system name you create in the 2D environment in the 3D **Workspace Explorer**.
- Do not delete all the components under the parent system, you must leave at least one component undeleted to see the changes made in the 2D environment. For example, if you delete all the objects under a parent system in the 2D environment, the changes will not be reflected in 3D environment. In a 3D environment, the parent system must always have at least one object.
- The **Place Belt Components** command supports the Add, Modify, and Delete modes only when you have at least one object left under the parent system.
- To delete a parent system, you must delete the parent system in the 3D environment. Then, the changes are reflected in the 2D environment.
- Material Handling symbols are delivered in the symbols share in the *[Reference Data Product Folder]*\MaterialsHandling\2D\Symbols folder.

Place Along Tab (Sketch 2D Place Belt Components Dialog Box - Equipment)

The **Place Along** tab defines placement information for equipment.

Part

Part Class

Select the equipment part category.

Part Name

Select the part within the selected part class.

Placement Parameters

Count

Specify the quantity of parts to place.

Offset

Specify the offset of the first part from the start point defined in **Selection Mode.** If you want the first part to be on the start point, use an offset of **0**.

Spacing

Specify the distance between each part.

Orientation

Select the orientation of the part to the belt. **Normal to Belt** is the default option. Orientation is unavailable until **Change** is clicked. You can also choose **Horizontal right**, **Horizontal left**, **Vertical up**, **Normal to line**, **Angle from line**, **Vertical Down**, and **Key in angle**.

- **Normal to line** - Select a line in the graphic view of the drawing. The equipment is oriented normal to the line.
- **Angle from line** - Type an angle, then select a line in the graphic view of the drawing. The equipment is oriented at the angle from the line normal.
- **Key in angle** - Type an angle. The equipment is rotated at the angle from the default orientation of the part.

Go

Click to place part symbols on the data layer.

NOTE **Go** is unavailable until a start point is selected in **Selection Mode**.

Change

Click to make **Orientation** available.

Initial Position and Direction

Belt segment endpoint

Select one of the belt arc or line segments. The closest end on the selected segment is the start point.

Smartmouse point

Select a point on the belt profile using Sketch 2D relationships. For example, this is useful for selecting the intersection of the belt with construction geometry. For more information, see *Draw with relationships* in *Sketch 2D User's Guide*.

Left to right

Places part symbols from left to right in the drawing view, starting at the point selected in **Selection Mode**.

Right to left

Places part symbols from right to left in the drawing view, starting at the point selected in **Selection Mode**.

Place Along Tab (Sketch 2D Place Belt Components Dialog Box - Modules)

The **Place Along** tab defines placement information for modules.

Part

Part Class

Select the module category.

Part Name

Select the part within the selected part class.

Placement Parameters

Count

Specify the quantity of module segments to place.

Offset

Specify the offset of the first module from the start point defined in **Selection Mode.** If you want the first module to be on the start point, use an offset of **0**.

Spacing

Specify the distance between each module segment. The default value is set by the module selected in the **Name** box. You usually do not change this value.

Idler Height

Specify the height of idlers being used with the belt, as defined using **Place Equipment**  . The module is offset from the belt by this height.

Go

Click to place module symbols on the data layer.

NOTE **Go** is unavailable until a start point is selected in **Selection Mode**.

Initial Position and Direction

Belt segment endpoint

Select one of the belt arc or line segments. The closest end on the selected segment is the start point.

Smartmouse point

Select a point on the belt profile using Sketch 2D relationships. For example, this is useful for selecting the intersection of the belt with construction geometry. For more information, see *Draw with relationships* in *Sketch 2D User's Guide*.

Left to right

Places symbols from left to right in the drawing view, starting at the point selected in **Selection Mode**.

Right to left

Places symbols from right to left in the drawing view, starting at the point selected in **Selection Mode**.

Place Tab (Sketch 2D Place Belt Components Dialog Box - Miscellaneous)

The **Place** tab defines placement information for miscellaneous components.

Part

Part Class

Object is the only option.

Part Name

ControlPoint is the only option.

Control Point Properties

Type

Type of control point. Select **Control Point**, **Key Point**, or **Insertion Point**.

Subtype

Point subtype. Select **Process Equipment**, **Mechanical Equipment**, **Foundation**, **Structure**, **Elevation Callout**, or **Ad Hoc Note**.

Name

Type a point name. The value of **Name** is used for the control point when it is imported into the 3D model. If **Name** is blank, the naming rule in the 3D model assigns a name.

Layer Properties Dialog Box

Provides options for managing layers and parent systems for symbols. You can access this dialog box by selecting **New Layer** or **Properties** on the *Configure Tab (Sketch 2D Place Belt Components Dialog Box)* (on page 165).

Layer Name

Specifies the name of the drawing layer. Type a name in the box, or select a name in the list. Only layers that are created using **New Layer** appear in the list.

3D System

Specifies the name of the system used as the parent after symbol placement in the 3D model. If the system does not exist, the software creates a new generic system and assigns the specified name to it. For more information, see *New Generic System* in the *System and Specifications User's Guide*.

OK

Accepts the options and closes the dialog box.

Cancel

Closes the dialog box without accepting the options.

Belt Correction Tool (Sketch 2D - Custom Command)

Checks for any faults in a belt profile. You can check for multiple connections, non-belt segments, and missing continuity and constraints in belt segments in **Sketch 2D**.

NOTE The belt profile must be grouped to use this command.

Belt Correction Tool Dialog Box (Sketch 2D Custom Command) (on page 171)

Check belt profile using Belt Correction Tool.

1. In **Sketch 2D**, click **Belt Correction Tool** .

   *The **Belt Correction Tool** dialog box appears.*
2. Select the belt profile, and click **Check**.

   *The **Belt Correction Tool** dialog box displays the available test criteria and their statuses.*
3. Select the option that needs correction.

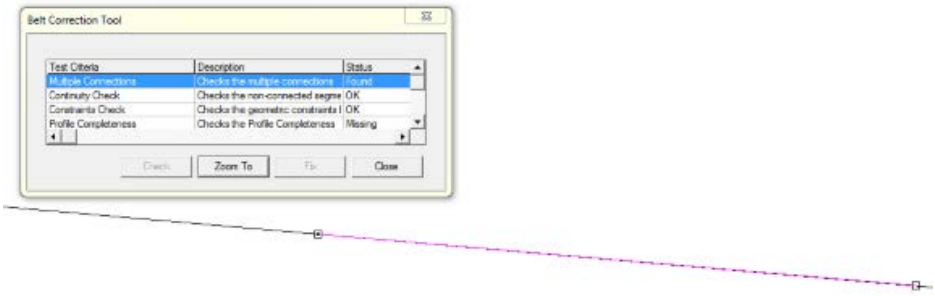
   For example:

   a. Select **Multiple Connections** if the status is **Found**.

      *The **Zoom To** option becomes available.*

   b. Click **Zoom To**.

      The software zooms to the location of the duplicate element.



4. Delete the duplicate element, and save the profile.
5. Click **Close**.

   *The **Belt Correction Tool** dialog box closes.*
6. Click **Accept**.
7. Click **Finish**.

   The software corrects the belt profile.

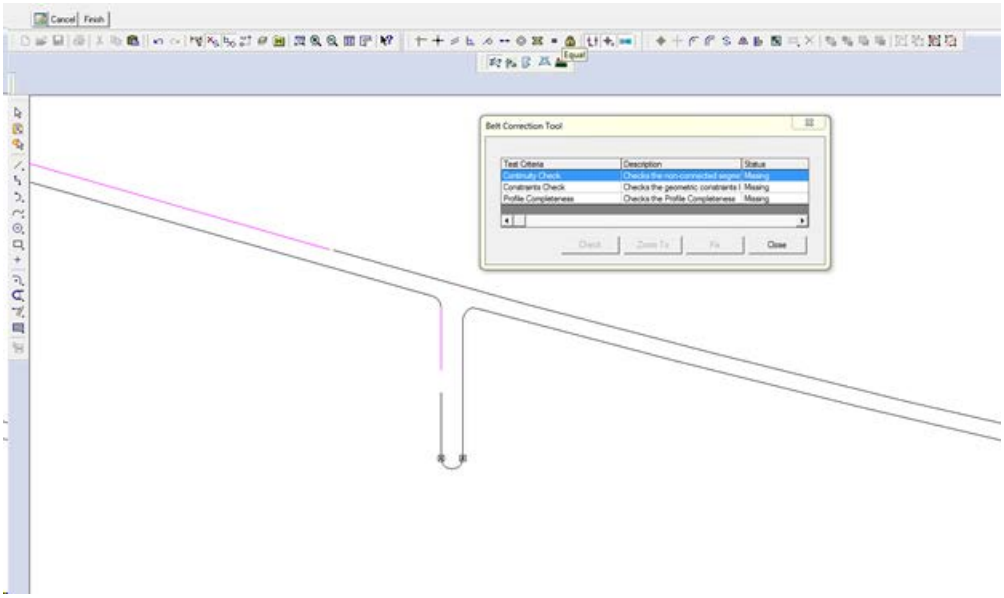
NOTE For options such as **Continuity Check**, **Multiple Connections**, and **Non-Belt Segments**, you must correct the faults manually and use the **Belt Correction Tool** command again to confirm if the status is **OK**.

Belt Correction Tool Dialog Box (Sketch 2D Custom Command)

Sets options for checking and fixing a belt profile.

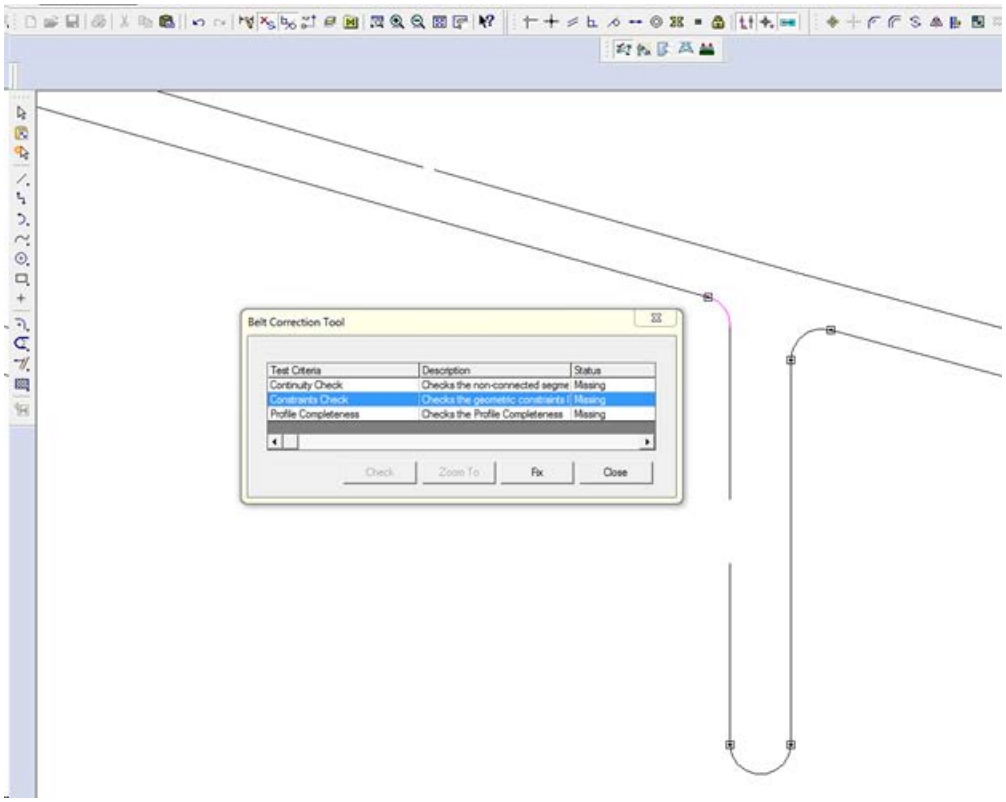
Continuity Check

Checks for continuity in the belt segments. If the status is **Missing**, you must correct them manually. When you select **Continuity Check**, the software highlights the belt segments that have the continuity missing.



Constraints Check

Checks for constraints in the belt segment. If the status is **Missing**, select **Constraints Check** to have the software highlight the belt segments that have the missing constraints. Click **Fix** to create constraints.

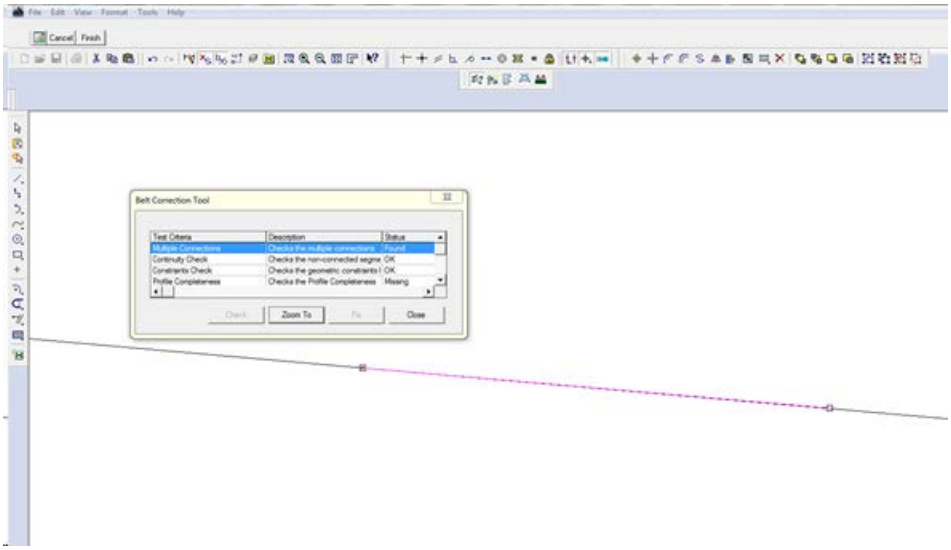


Profile Completeness

Checks if the profile is continuous and has constraints with no multiple connections and non-belt segments. You can place the belt profile when the status is **OK**.

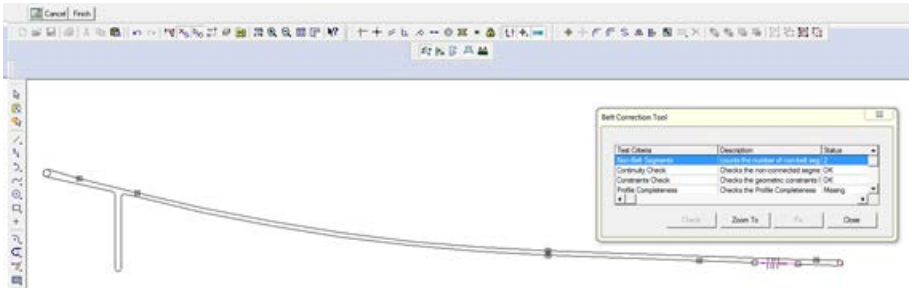
Multiple Connections

Checks for multiple connections. If the status is **Found**, you must correct them manually. When you select **Multiple Connections**, the software highlights the belt segments that have multiple connections. When you delete the missing connection or non-belt segment, you must also manually delete the relationship constraints at that location.



Non-Belt Segments

Checks for non-belt segments. If found, you must remove them manually. The status displays the number of extra segments in the group that are not related to the belt profile. When you select **Non-Belt Segments**, the software highlights the belt segments that have non-belt segments.



Check

Checks the bet profile.

Zoom To

Adjusts the display of the model so that the highlighted problem area displays.

Fix

Corrects the selected option for the belt profile.

Close

Closes the dialog box.

Place 2D Chute Shapes (Sketch 2D - Custom Command)

Places chute shapes symbols in Sketch 2D. The symbols are then written to the Material Handling 3D model.

Place 2D Chute Shapes Dialog Box (on page 174)

Place 2D Chute Shapes Dialog Box

Controls parameters for placing 2D chute shapes.

Symbol Directory

Specifies the folder that contains the symbol file. Click **...** to browse to the folder.

Select 2D Symbols

Specifies the file that defines the 2D symbol.

Preview

Displays an example diagram of the 2D symbol. The attributes defined on the diagram are available for you to edit in the **Attribute Viewer**. Click **Symbol Explorer** to display the **Attribute Viewer**.

Mirror Shapes

Mirror Head Section

Controls whether the head section is mirrored. If you select this option, the software mirrors the head section about the belt path.

Mirror Transition Section

Controls whether the transition section is mirrored. If you select this option, the software mirrors the transition section about the belt path.

Mirror Bottom Section

Controls whether the bottom section is mirrored. If you select this option, the software mirrors the bottom section about the belt path.

Rotation Angle

Angle of Rotation for Transition Section

Defines the angle of rotation for transition section. Type the required value in degrees.

Angle of Rotation for Bottom Section

Defines the angle of rotation for bottom section. Type the required value in degrees.

Belt Position

Modify Belt position

Specifies whether or not to modify the belt position. Select this option to enable the Origin

and Destination offset options.

Origin X Offset

Specifies the original X offset for the belt.

Origin Y Offset

Specifies the original Y offset for the belt.

Destination X Offset

Specifies the resulting X offset for the belt.

Destination Y Offset

Specifies the resulting Y offset for the belt.

Create Trajectory (Sketch 2D - Custom Command)

Places trajectory paths along the material flow. The software adjusts the trajectory shape based on the conveyor type. **Create Trajectory** is a custom command in the Sketch 2D environment.

The MHCreateTrajectory.ocx file is delivered by default to the *[Product Folder]*\MaterialsHandling\Client\Bin\ folder.

Create Trajectory Dialog Box (on page 175)

Create Trajectory Dialog Box

Controls settings for adding trajectory paths to the model.

Belt Capacity

Specifies the capacity for the belt.

Belt Width

Specifies the width of the belt.

Belt Speed

Specifies the speed of the belt.

Belt Thickness

Specifies the thickness of the belt.

Surcharge Angle

Specifies the dynamic angle of repose of the stockpile.

Troughing Angle

Specifies the troughing angle for the belt.

Density of Material

Specifies the density of the material carried by the belt.

Pulley Diameter

Specifies the diameter of the pulley.

Type of Conveyor

Specifies the conveyor type. You specify the angle for belts that are not horizontal.

Height of Pulley Center from Ground

Specifies the height of the pulley center.

Plot Trajectory Towards Left

Indicates that the trajectory plots towards the left.

Plot Trajectory Towards Right

Indicates that the trajectory plots towards the right.

Place 2D Trestle (Sketch 2D - Custom Command)

 Places trestle symbols in Sketch 2D. The symbols are then written to the Material Handling 3D model.

Place 2D Trestles Dialog Box (on page 176)

Layer Properties Dialog Box (on page 179)

Place 2D Trestles Dialog Box

Controls parameters for placing 2D trestles.

- *Place Tab (2D Trestles Dialog Box)* (on page 176)
- *Columns and Top Beam Tab (2D Trestles Dialog Box)* (on page 178)
- *Bracing and Cross Beams Tab (2D Trestles Dialog Box)* (on page 179)

Place Tab (2D Trestles Dialog Box)

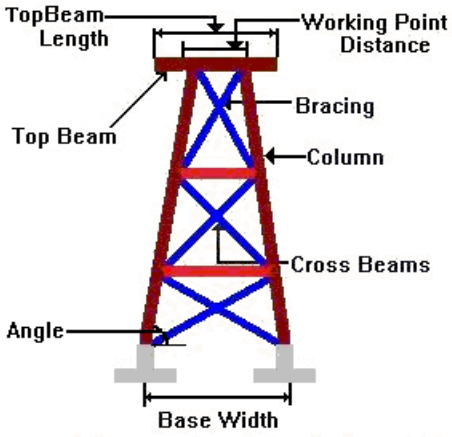
Controls parameters for placing 2D trestles.

Trestle Type

Select the trestle type from the list.

Preview

Displays an example diagram similar to the figure below:



Symbol Details

Symbol Directory

Specifies the folder in which the symbol file resides. Click **...** to browse to the folder.

Symbol File

Specifies the file that defines the symbol.

Placement Parameters

Base Width

Specifies the trestle base width.

Top Beam Length

Specifies the trestle top beam length.

Clearance Height

Specifies the height between the first selection point for trestle 2D symbol placement and the top of the top beam.

Display Grid Lines

Displays the grid lines and planes in the model and in the **Workspace Explorer**. This allows you to use the grid planes to modify the trestle members in the model.

Number of Cross Beams

Specifies the number of cross beams included in the trestle. This option is only available if the **Trestle Type** is set to **Trestle**.

Angle

Indicates the angle between the column and the base. Type the angle in the box. This option is only available if the **Trestle Type** is set to **Trestle**.

Working Point

Specifies the top distance between the columns. Type the working point distance in the box.

This option is only available if the **Trestle Type** is set to **Trestle**.

Footings

Specifies the required footings. Type a name in the box, or select a footing from the **Select Equipment** dialog box. This option is only available if you select the check box.

Depth

Specifies the depth of the footings. Type the depth in the box.

Layer Name

Specify the required layer. Type a name in the box, or select a name in the list. To create a new layer, click **New Layer**.

New Layer

Opens the **Layer Properties** *dialog box* (on page 179) so you can create a new layer. Layers created using **New Layer** are also visible in the **Layer Display** dialog box, which is accessible using the **Tools > Layers** command in Sketch 2D. For more information, see the Sketch 2D Help.

Properties

Opens the **Layer Properties** dialog box so you can specify the name of the parent system.

Columns and Top Beam Tab (2D Trestles Dialog Box)

Controls parameters for the trestle columns and top beams.

Member Type

Specifies the member type associated with the **Columns** or **Top Beam**.

Standard

Specifies the standard associated with the **Columns** or **Top Beam**.

Section

Specifies the section shape associated with the **Columns** or **Top Beam**.

Material

Specifies the material associated with the **Columns** or **Top Beam**.

Grade

Specifies the material grade associated with the **Columns** or **Top Beam**.

Cardinal Point

Specifies the cardinal point associated with the **Columns** or **Top Beam**.

Angle

Specifies the angle associated with the **Columns** or **Top Beam**.

Bracing and Cross Beams Tab (2D Trestles Dialog Box)

Controls parameters for the trestle bracing and cross beams.

Bracing Type

Specifies the bracing type associated with the **Bracing**.

Member Type

Specifies the member type associated with the **Bracing**.

Standard

Specifies the standard associated with the **Bracing** or **Cross Beams**.

Section

Specifies the section shape associated with the **Bracing** or **Cross Beams**.

Material

Specifies the material associated with the **Bracing** or **Cross Beams**.

Grade

Specifies the material grade associated with the **Bracing** or **Cross Beams**.

Cardinal Point

Specifies the cardinal point associated with the **Bracing** or **Cross Beams**.

Angle

Specifies the angle associated with the **Bracing** or **Cross Beams**.

Member

Specifies the member type associated with the **Cross Beams**.

Layer Properties Dialog Box

Provides options for managing layers and parent systems for symbols. You can access this dialog box by selecting **New Layer** or **Properties** on the *Place Tab (2D Trestles Dialog Box)* (on page 176).

Layer Name

Specifies the name of the drawing layer. Type a name in the box, or select a name in the list. Only layers that are created using **New Layer** appear in the list.

3D System

Specifies the name of the system used as the parent after symbol placement in the 3D model. If the system does not exist, the software creates a new generic system and assigns the specified name to it. For more information, see *New Generic System* in the *System and Specifications User's Guide*.

OK

Accepts the options and closes the dialog box.

Cancel

Closes the dialog box without accepting the options.

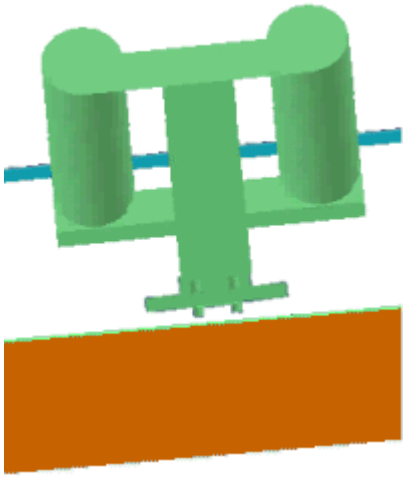
Idler Packing Thickness (Sketch 2D - Custom Command)

Places additional packing material thickness below idlers so that they mount perfectly on the trusses. The software automatically calculates the required packing thickness for the 3D idler symbols in the model. The packing thickness can be different for each idler.

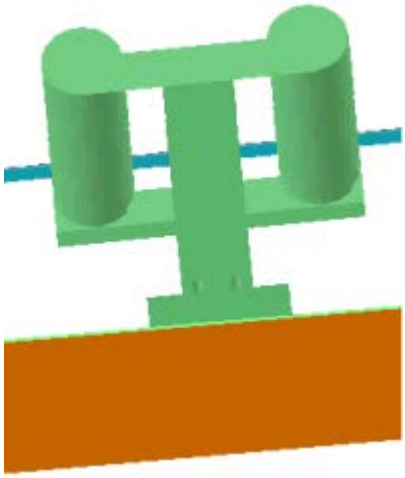
NOTE A packing plate is a distinct object type in the model and has no relationship to plates created in the Molded Forms task.

Additional packing is generally required when the belt transitions between a linear and curved profile and the distances vary between idlers and trusses.

Before packing:



After adding packing:



Reports

Use the **Tools > Run Reports** command to generate an idler packing thickness report. For more information about reports, see *Run Report* in the *Common User's Guide*.

Idler Packing Thickness Ribbon

Select Truss

Specifies the truss to associate with idlers.

Add/Remove

Adds or removes the selected idlers. No packing is added to a removed idler.

NOTE You should only add or remove idlers associated with the selected truss. Do not add idlers from other trusses.

Accept

Sets packing thickness attributes on the idlers.

Cancel

Removes the packing thickness attributes from the idlers.

Close

Exits the command.

What do you want to do?

- *Add idler packing thickness* (on page 181)
- *Remove idler packing thickness* (on page 182)
- *Create an idler packing thickness report* (on page 183)

Add idler packing thickness

1. Click **Place Belt Component** on the vertical toolbar.
2. Click **Modify or Delete Belt Components**.

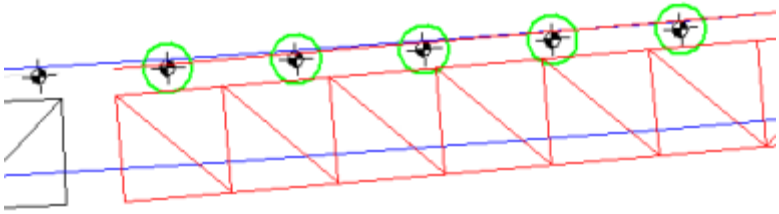
   *The **Review** dialog box appears.*
3. With **Select Mode** set to **Modification**, select a conveyor from the grid, and click **OK**.
4. Click **Sketch 2D** on the ribbon.

   Smart 3D displays the message, "Rad 2D file is updated according to the 3D changes."
5. Click **OK**.

   The 2D environment opens, showing the symbols for trusses, idlers, and the belt profile.
6. Click **Idler Packing Thickness**.

   *The **Idler Packing Thickness** ribbon appears.*
7. Select a truss symbol.

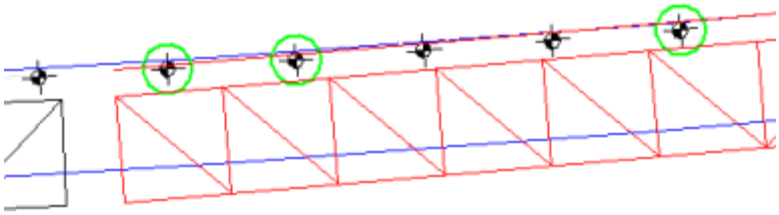
   The idler symbols are selected and circled in green.



8. Click **Add/Remove**.

9. Select idlers to remove as required.

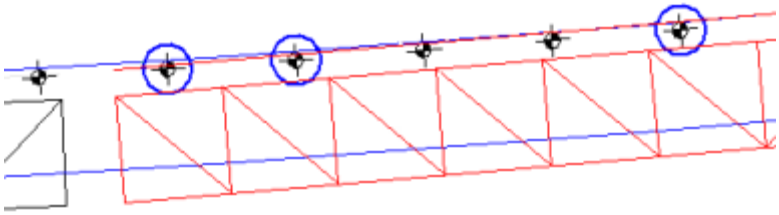
   The green circle is removed when an idler is not selected.



NOTE Select a removed idler again to add it.

10. Click **Accept** ✓ to set the packing thickness on the selected idlers.

    The selected idlers are now circled in blue.



11. Click **Close**.
12. Click **Finish**.

    The Sketch 2D environment closes and the software returns to the 3D environment.

13. Click **Finish** on the **Place Belt Component** ribbon.

    Packing is added between each idler and the truss.

Remove idler packing thickness

1. Click **Place Belt Component** on the vertical toolbar.
2. Click **Modify or Delete Belt Components**.
3. With **Select Mode** set to **Modification**, select a conveyor with existing packing on the idlers.
4. Click **OK**.
5. Click **Sketch 2D** on the ribbon.

   Smart 3D displays the message, "Rad 2D file is updated according to the 3D changes."

6. Click **OK**.

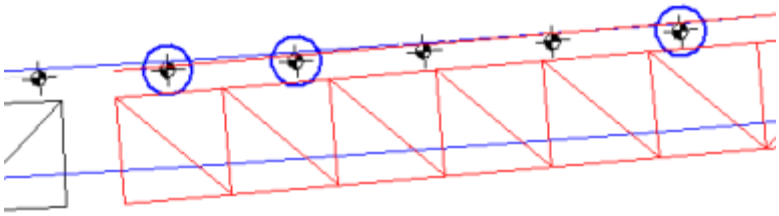
   The 2D environment opens, showing the symbols for trusses, idlers, and the belt profile.

7. Click **Idler Packing Thickness**.

   *The **Idler Packing Thickness** ribbon appears.*

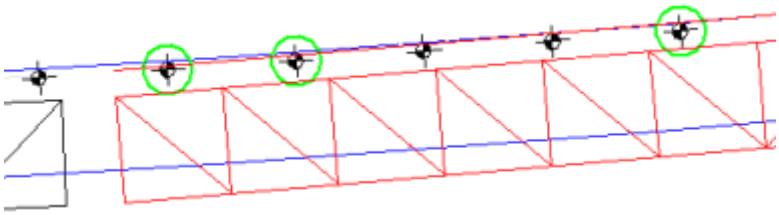
8. Select a truss symbol with existing packing on the idlers.

   The idler symbols are circled in blue.



9. Click **Cancel** ✕ to remove the packing thickness.

   The idlers are now circled in green.



10. Click **Close**.
11. Click **Finish**.

    The Sketch 2D environment closes and the software returns to the 3D environment.

12. Click **Finish** on the **Place Belt Component** ribbon.

    Packing is removed between each idler and the truss.

Create an idler packing thickness report

1. Click **Tools > Run Reports**.
2. On the **Catalog Reports** tab, expand the tree view, and select **Materials Handling**.
3. Select **Materials Handling MHE_PackingThickness_Reports** in the list of available report templates.
4. Click **Run**.

   *The **Filter Properties for Asking Filter** dialog box appears.*

5. Select a system containing the idlers with packing needed for the report, and click **OK**.

   The dialog box closes, and the report creation process begins. When processing completes, the software opens the report so that you can view its contents.

APPENDIX A

Appendix: Belt Components Catalog Exporter

MHCatalogExporter.exe is an administration tool used to configure the catalog items available with **Place Belt Components** (Sketch 2D custom command). The tool is delivered in the *[Product Folder]*\Materials Handling\Tools\bin folder. to access the tool click **All Programs** > **Intergraph Smart 3D** > **Database Tools** > **Belt Components Catalog Exporter** .

Belt Components Catalog Exporter exports equipment part names, module part names, wearplate part names, part numbers, and other details from the active catalog to a list file. The list file is used by **Place Belt Components** to link the 2D symbols to their corresponding 3D catalog item.

Catalog Exporter Dialog Box (on page 186)

Structure to Export Dialog Box (on page 187)

Equipment to Export Dialog Box (on page 188)

Wearplates to Export Dialog Box (on page 189)

What do you want to do?

- *Export equipment from the catalog* (on page 185)
- *Export modules from the catalog* (on page 185)
- *Export wearplates from the catalog* (on page 186)

Export equipment from the catalog

1. Click **Start > All Programs > Intergraph Smart 3D > Database Tools > Belt Components Catalog Exporter.**

   *The **Catalog Exporter** dialog box displays.*
2. In the **Catalog Name** box, select a catalog.
3. In the **Select Catalog Object Type** list, select **Materials Handling Equipment**.
4. In the **Select catalogs items to export** box, select one or more equipment items.
5. In the **List File** box, type a path and file name, or click **Browse**.
6. Click **Export**.

   *The **Equipment to Export** dialog box displays.*
7. In the **Export** column, select the equipment items to export.
8. Click **OK**.

   *After the list file is created, **View File** is active in the **Catalog Exporter** dialog box.*
9. Click **View File**.

   The list file opens.

Export modules from the catalog

1. Click **Start > All Programs > Intergraph Smart 3D > Database Tools > Belt Components Catalog Exporter.**

   *The **Catalog Exporter** dialog box displays.*
2. In the **Catalog Name** box, select a catalog.
3. In the **Select Catalog Object Type** list, select **Materials Handling Modules**.
4. In the **Select catalogs items to export** box, select one or more module items.
5. In the **List File** box, type a path and file name, or click **Browse**.
6. Click **Export**.

   *The **Structure to Export** dialog box displays.*
7. In the **Export** column, select the structure items to export.
8. Click **OK**.

   *After the list file is created, **View File** is active in the **Catalog Exporter** dialog box.*
9. Click **View File**.

   The list file opens.

Export wearplates from the catalog

1. Click **Start > All Programs > Intergraph Smart 3D > Database Tools > Belt Components Catalog Exporter.**

   *The **Catalog Exporter** dialog box displays.*
2. In the **Catalog Name** box, select a catalog.
3. In the **Select Catalog Object Type** list, select **Materials Handling Wearplates**.
4. In the **Select catalogs items to export** box, select one or more wearplate items.
5. In the **List File** box, type a path and file name, or click **Browse**.
6. Click **Export**.

   *The **Wearplates to Export** dialog box displays.*
7. In the **Export** column, select the wearplate items to export.
8. Click **OK**.

   *After the list file is created, **View File** is active in the **Catalog Exporter** dialog box.*
9. Click **View File**.

   The list file opens.

Catalog Exporter Dialog Box

Specifies equipment, module, or wearplate catalog items to export from the catalog into a list file.

Databases

Server Name

Displays the active server. This is a read-only field.

Site Name

Displays the active site database name. This is a read-only field.

NOTE The active server and site database are defined on a user-account basis using the **Modify Database and Schema Location** and **Database Wizard** tools. For more information, see *Utilities and Services* in the *Project Management User's Guide*.

Catalog Name

Specifies a catalog database associated with the active site database.

Catalog Browser

Select Catalog Object Type

Specifies the type of objects available for export. Available options are:

- **Materials Handling Equipment** displays Material Handling equipment items in the catalog database. The items display in the **Select catalog items to export** box.

- **Materials Handling Modules** displays Material Handling modules items in the catalog database. The items display in the **Select catalog items to export** box.
- **Materials Handling Wearplates** displays Material Handling wearplates items in the catalog database. The items display in the **Select catalog items to export** box.

Select catalog items to export

Specifies catalog items to export to the list file. Displayed catalog items are based on the selection of **Materials Handling Equipment, Materials Handling Modules,** or **Materials Handling Wearplates** in the **Select Catalog Object Type** list**.**

List File

Specifies the path and name for the list file. Type the path, or click **Browse** to select a path.

Browse

Opens a dialog box to select a path and name for the list file.

Export

Exports selected catalog items to the defined list file.

View File

Opens the list file. This option is available when you select **Export.**

Close

Closes the dialog box.

Structure to Export Dialog Box

Specifies truss structure items to export from the catalog into a list file. The dialog box appears when you select **Materials Handling Modules** and click **Export** on the **Catalog Exporter** dialog box. For more information, see *Catalog Exporter Dialog Box* (on page 186).

Select All

Exports all items.

Export

Exports the item.

Part Class Name

Displays the truss category.

Part Name

Displays the specific truss part name.

Symbol Name

Displays the symbol file used by the truss part. If needed, type a different symbol file name. The symbol must be in the .SYM format.

Type

Specifies the type of structure. The default setting is **0**. Setting this value to **1** indicates a truss.

Subtype

Specifies the type of truss.

- **0** - Linear truss
- **1** - Convex truss
- **2** - Concave truss

Origin

Specifies the placement point for the truss.

- **0** - Left
- **1** - Center
- **2** - Right

Length

Displays a default value for truss length. Type the needed length.

Radius

Displays a default value for the truss radius. For a linear truss, the value is **0**. For convex and concave trusses, type the required value.

OK

Exports the data to the list file.

Cancel

Closes the dialog box without exporting the truss structure items.

Equipment to Export Dialog Box

Specifies Material Handling equipment items to export from the catalog into a list file. The dialog box appears when you select **Materials Handling Equipment** and click **Export** on the **Catalog Exporter** dialog box. For more information, see *Catalog Exporter Dialog Box* (on page 186).

Select All

Exports all items.

Export

Exports the item.

Part Class Name

Displays the equipment part class.

Part Name

Displays the specific part name.

Symbol Name

Displays the symbol file used by the equipment part. If needed, type a different symbol file name. The symbol must be in the .sym format.

Belt Offset

Displays a default value for belt offset. Type the needed length.

Wearplates to Export Dialog Box

Specifies Material Handling wearplates items to export from the catalog into a list file. The dialog box appears when you select **Materials Handling Wearplates** and click **Export** on the **Catalog Exporter** dialog box. For more information, see *Catalog Exporter Dialog Box* (on page 186).

Select All

Exports all items.

Export

Exports the item.

Part Class Name

Displays the wearplates specific part class name.

Part Name

Displays the specific part name.

Symbol Name

Displays the symbol file used by the wearplates part. If needed, type a different symbol file name. The symbol must be in the .sym format.

No. of Bolts

Displays the number of bolts available on the wearplates.

Plate Width

Displays the width of the wearplates in mm.

Plate Height

Displays the height of the wearplates in mm.

Horizontal Bolt

Displays the horizontal spacing between the two bolts of the wearplates in mm.

Vertical Bolt

Displays the vertical spacing between the two bolts of the wearplates in mm.

APPENDIX B

Appendix: Symbol Directory

MHSetSymbolDirectory.exe is an administration tool used to configure the file path for the symbol folder for all Sketch 2D and Sketch 2D commands. The tool is delivered in the *[Product Folder]*\Materials Handling\Tools\bin folder. If you do not use **Symbol Directory**, the folder path for the symbol folder is the default location, *[Product Folder]*\SharedContent\MaterialsHandling\2D\Symbols.

Change the folder path of the Symbol Directory

1. Double-click *[Product Folder]*\Materials Handling\Tools\bin**MHSetSymbolDirectory.exe**.

   *The **SetSymbolDirectory** dialog box appears.*

2. Type the target folder path, or click **Browse**.
3. Click **OK**.

   The folder path for the symbol folder is automatically updated for all Sketch 2D and Sketch2D commands.

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